Energy, Mines and Resources Canada | Énergie, Mines et Ressources Canada

CANMET

Canada Centre for Mineral and Energy Technology | Centre canadien de la technologie des minéraux et de l'énergie

OFFSHORE STRUCTURES: THE DESIGN AND ANALYSIS BY THE FINITE ELEMENT METHOD

R. BELL
ENGINEERING & METAL PHYSICS SECTION

This document is an unpublished report
for limited circulation
to
reports
may be cited
the author
of CANMET
Energy, Mines and Resources Canada (CANMET)

MAY 1982

ENERGY RESEARCH PROGRAM

115 pp

PHYSICAL METALLURGY RESEARCH LABORATORIES
REPORT: ERP/PMRL 82-41(TR)

PHYSICAL METALLURGY RESEARCH LABORATORIES
REPORT ERP/PMRL 82-41(TR)

OFFSHORE STRUCTURES: THE DESIGN AND ANALYSIS BY THE FINITE ELEMENT METHOD

by

R. Bell*

ABSTRACT

The published literature on the application of the finite element method (FEM) to the design and analysis of offshore structures is reviewed. The most commonly used commercially available FE packages are identified and their capabilities for the analysis of offshore structures are discussed.

A selected list of references is included together with a list of computer bureaux where these FE packages may be accessed.

---

*Carleton University, Engineering Department, Ottawa, Ontario.

LABORATOIRES DE RECHERCHE EN MÉTALLURGIE PHYSIQUE

RAPPORT PRE/LRMP 92-41(TR)

STRUCTURES OFFSHORE: DESIGN ET ANALYSE PAR LA MÉTHODE DES ÉLÉMENTS FINIS

par

R. BELL

RESUME

On présente une revue de la littérature sur l'application de la méthode des éléments finis au design et à l'analyse des structures offshore. On identifie les programmes commerciaux d'éléments finis les plus utilisés et on évalue leur performance pour l'analyse des structures offshore.

Une bibliographie est incluse ainsi q'une liste des bureaux de calcul ou l'on peut avoir accès à ces programmes commerciaux.

CONTENTS

i Abstract and ii Résumé

1. Introduction 1

1.1 Background 1

1.2 Description of Offshore Structures 2

Figures 8

2. Design of Offshore Structures 15

2.1 Introduction 15

2.2 Static Analysis 18

2.3 Dynamic Analysis 18

2.4 Stress Analysis of Joints 20

2.4.1 Finite Element Method 23

2.4.2 Acrylic Models 23

2.4.3 Photoelasticity 24

2.4.4 Full Size Steel Nodes 24

2.4.5 Parametric Equations 24

2.5 Fatigue Analysis 25

2.6 Crack Propagation 29

Figures 34

3. General Purpose Finite Element Programs for the Analysis of Offshore Structures. 45

3.1 General Review of FEM 45

3.2 General Purpose Finite Element Programs 50

3.3 Computer Packages for the Analysis of Offshore Structures 51

3.3.1 Det Norske Veritas 51

3.3.2 Lloyd's Register of Shipping 53

3.3.3 American Bureau of Shipping 57

3.3.4 SACS 59

3.3.5 ASAS 62

3.3.6 MARCS 66

3.3.7 Others 69

3.4 Program Availability 69

Figures 77

4. Application of FE Programs to the Analysis of Offshore Structures ... 86
4.1 Introduction 86
4.2 DNV/SESAM69 86
4.2.1 Analysis of Joints 86
4.2.2 Analysis of CONDEEP Platform 86
4.3 Lloyd's/NASTRAN 87
4.3.1 Dynamic Analysis 87
4.3.2 Analysis of Stiffened Joints 88
4.4 ASAS 89
4.4.1 Analysis of Joints 89
4.4.2 Dynamic and Fatigue Analysis 89
4.5 Lauch Analysis Comparison 91

Figures 92

5. Conclusion 96

6. References 97

APPENDIX I 108

1. INTRODUCTION

1.1 BACKGROUND

The National Energy Program (NEP) created incentives to promote increased activity in the exploration of the hydrocarbon resources situated off the Canadian coast (1). To exploit these offshore resources large and complex structures are required, many of which will be designed and constructed on the basis of the experience gained during the past fifteen years in the North Sea. These structures will operate in some of the most hazardous coastal waters in the world and therefore in order that they continue to meet their design and operational requirements, an ongoing program of research and development in design, material selection, construction, inspection, maintenance and operation is required.

This report is written to review one aspect of the aforementioned research, namely the use of the Finite Element Method (FEM) in the design and stress analysis of complex offshore structures. In particular, the capabilities of commercially available general purpose finite element packages, as they pertain to the design and analysis of offshore structures, are highlighted.

This report is the initial phase of a longer term goal; namely the development of a comprehensive "Fracture Control Plan", the purpose of which, is to minimize the risk of failure in these complex structures.

The report is organized as follows. In section 1.2 of this chapter a brief history of the development of offshore platforms, and a short description of the variety of offshore exploration and production units now in service or proposed, is given. In Chapter 2 an outline of the various types of analyses of offshore structures which may be undertaken using FEM are described. Chapter 3 includes a brief outline of the FEM followed by descriptions of the more widely used, commercially available, general purpose FE packages with specific capabilities for the analysis of offshore structures. Chapter 4 gives a limited review of examples from the literature where these packages have been used in the analysis of offshore structures.

Included in this report is a list of computer bureaux where these FE packages may be accessed, a list of addresses of the developers of these computer codes and a limited list of references.

1.2 DESCRIPTION OF OFFSHORE STRUCTURES

The initial structures designed for offshore oil exploration and development were tubular steel structures with four legs designed for the shallow waters of the Gulf of Mexico. The first fixed offshore structure was installed in 7 m of water in the Gulf of Mexico in the late 1940's. Since then there has been steady development in that area, with over 25 fixed platforms installed each year. The next major offshore venture occurred in the North Sea in the 1960's and the first fixed steel platform was installed in 22 m of water in 1966. Designers began to learn of the harshness of the North Sea environment by observing the platform and subsequent installations in deeper water. At this stage the design emphasis swung away from failure due to overload, to failure from fatigue, thus starting a major research effort on the fatigue behaviour of tubular structures (2,3).

Recently, with the extension of exploration and production into deeper water of the North Sea and the Gulf of Mexico, (300 m), structures have become more flexible and an analysis of the structures with regard to dynamic response had to be included (4). Today, with plans to exploit the hydrocarbon resources in still deeper waters (500 to 700 m) many innovative designs are being discussed including tension leg platforms and guyed tower platforms.

At present offshore explorations for hydrocarbon resources has spread worldwide from the Gulf of Mexico and North Sea to the Canadian Arctic, the coasts of Africa, South America, S.E. Asia and New Zealand. To carry out this offshore exploration and production many different types of offshore structures have been designed and built.

Offshore exploration and production units can be divided into the two main groups of mobile and fixed structures as shown in Table 1.1 and a brief description of some of the many variations in design follow.

Drilling Ships

Drilling ships are established as economical and versatile exploration units that have the advantage of rapid transit between drilling stations. However, the ships' motion and anchor capacity place limits on the weather conditions in which they can operate. This class of exploration unit has been used extensively in the Canadian Arctic and offshore Labrador. Drilling ships are normally not considered as offshore structures but as ships and will therefore not be discussed further in this report.

Jack-up Platforms

Jack-up barges, Fig. 1.1, are the most economical exploration units for drilling in shallow waters. Today, these units are capable of drilling in water depths of 100 m. The main disadvanatage of these rigs is that movement between drilling stations is slow and jacking up and down must be performed in good weather. The critical area of these structures is the legs and therefore the leg sections and elevation systems require sophisticated design and construction procedures. High strength steels (yield strength 700 MN/m^2) have been used in the lattice type construction of the legs to reduce weight. Fatigue is usually not a major problem with jack-up units because with regular movement of the barge different positions in the legs are highly stressed. It is also possible for the unit to return to dock periodically so that non destructive evaluation (NDE) of the legs may be carried out (5). In some cases where these rigs are used as early production systems and the rig is held for an extended time period at a single elevation, fatigue effects on the legs and support structure must be considered.

Semi-submersible Units

The semi-submersible drilling unit, an example of which is shown in Fig. 1.2, is a stable drilling or production platform. These rigs can have three or more large-diameter tubular legs (~10 m diameter) supporting the deck. The pontoons under the legs provide buoyancy during tow-out and are flooded to stabilize the rig in its drilling position where it is held on station by multiple anchors. In shallow waters, less than 30 m, the rig can sit on the sea bottom. The main advantages of such a mobile rig are its greatly increased stability in comparison to a drilling ship and its durability under severe winter conditions. However, a mobile rig is extremely expensive and it is slow to move since it has to be towed.

The main structural concerns with this type of unit are the welding and performance of the large tubular joints or nodes. A number of structural problems and failures with this class of rig have been documented in the literature (6,7,8).

Fixed Steel Platforms

Fixed steel platforms are the most widely used platforms in offshore production mainly because steel has a proven track record of design flexibility, ease of modification and repair, lightness, ductility and predictable and

controllable corrosion resistance. These structures shown in Fig. 1.3 are stable, and the design and analysis is now relatively straightforward enabling displacements and stresses to be predicted with good accuracy. The early jackets were of tubular space frame construction and had four legs that served as templates for piles that held the structure to the seabed. In deeper waters the number of legs, leg diameter and spacing had to be increased and extensive stiffening of the legs and braces was required. When the harsh environments and taller structures were taken into consideration a fatigue analysis including dynamic response had to be included in the design.

Many of these structures constructed with the main legs as flotation tanks, were towed to the site, positioned and then the construction was completed, Fig. 1.4. Today, with the advent of large, purpose built, transportation barges these large structures are completed in dry dock, carried to the production site in one piece, launched and then fixed to the seabed with piles. To date, the largest single-piece offshore jacket to be installed is in the Gulf of Mexico 176 km south-southeast of Galveston (9). This jacket is 290 m high and is anchored in 285 m of water. On final completion this structure will be 350 m high. This jacket was towed to the production site aboard a barge, launched and up-ended into position. It stands on eight 1524 mm diameter main tubular legs that have a 50 mm wall thickness.

The principal problems in the design and fabrication of fixed steel platforms are the tubular joints. In the structure described above there are 2000 such joints. The design of tubular joints may be governed by static strength but more likely by fatigue. The design, and static and fatigue behaviour of tubular joints has been the subject of many research projects and international conferences (2,3).

Concrete Gravity Structures

Concrete gravity structures have undergone rapid development during the past several years and a variety of designs are now in service. Two examples of the concrete gravity structures are the "Condeep" and "Seatank" designs, whose main features are shown in Figs. 1.5 and 1.6.

These structures sit on the seabed without the use of foundation piles and have oil storage capacity in the base. They have concrete substructures but the decks are built of steel to minimize weight. Concrete gravity structures have been installed in water depths of 150 m and generally have a base dimension of 100 m which would enable 1 million barrels of oil to

be stored. Typically a concrete gravity structure would contain 150,000 m^3 of concrete. The analysis, construction and installation of these structures is very involved (10,11,12).

The main advantage of these structures is that concrete is cheap, shapeable and of unlimited thickness. Also, the structure can be completed and fitted out prior to transportation to the production site. However, it should be noted that with high construction costs the total cost of this structure is greater than a comparable fixed steel platform. The corrosion resistance of concrete depends on many factors many of which are difficult to quantify and the removal and disposal of these structures after their useful life may cause some problems.

Steel Gravity Structures

Steel gravity platforms are also available, they are self-floating and can be transported, fully equipped, large distances to the production site (13). One design, shown in Fig. 1.7, consists of three flotation cylinders attached to a triangular base which supports a hexagonal lattice tower that in turn supports the deck. the cylinders are ballasted at the production location and the rig rests on the sea bed. The base width of this structure is approximately 80 m and the deck elevation is 96 m. Several of these structures were constructed in France and installed at production sites offshore of the Congo in water depths of 90 m.

Tension Leg Platform (TLP)

The tension leg platform (TLP) was developed for drilling and/or production in deeper waters (i.e. greater than 300 m). This unit is a semi-submersible floating structure, connected to anchors fixed to the seabed by vertical mooring lines, known as tension legs, as shown in Fig. 1.8. An excess of buoyancy provided by the bouyancy unit, which is greater than the platform weight, keeps the mooring lines under tension for all loading and weather conditions. The TLP is a permanently installed structure and its main advantages over tower platforms for deep water sites are cost and the fact that the fabrication, outfitting and commissioning of the equipment can be completed in inshore completion sites. The dynamic behaviour of the TLP is complex and is therefore receiving a lot of attention (14,15,16). The vertical motions (heave, pitch and roll) of these units are effectively restrained by the tension legs but the structure is free to yaw, surge and sway. These

motions and the variaton in load must be taken into account in the design of the foundations, tension legs, production equipment and general structure. A TLP is to be installed in the North Sea Hutton field in a water depth of 147 m in the near future (17).

Guyed Tower Platform (GTP)

Another recent design for deep water oil production that has received intense evaluation in recent years is the guyed tower platform (GTP), Fig. 1.9. Studies have indicated that the concept is technically feasible for water depths of 200 m to 700 m. Guyed towers are compliant structures and therefore limiting their dynamic response is of major concern. This is accomplished by ensuring that their natural period is very different from the period of the waves that carry a significant amount of wave energy.

Conventional fixed steel platforms, 100 m high, have natural periods of about 2s, which is less than the wave period of approximately 16s, so the dynamic amplification factor (DAF) is less than 1.0. As the height of the steel jacket structure increases to 300 m the natural period increases to 4-6s and the DAF is greater than 1, necessitating modifications in design to stiffen the structure. This results in heavier and more costly structures. Compliant structures, TLP and GTP are designed to have much higher natural periods of the order of 30-60s and therefore the dynamic amplification factor is again less than 1.0. These concepts are shown diagramatically in Fig. 1.10.

The proposed designs for guyed tower platforms consist of a tall uniform space frame tower, connected to the seabed by piles that support the deck. The tower is supported in the lateral direction by 16-32 guy wires connected to the seabed by anchors or piles at a radius of about 1000 m. A more complete description of this design concept is given in (18,19). The main advantages compared with conventional deep water steel jackets are less weight and less weather sensitivity; however, the main disadvantage is difficult installation in areas where there are strong tidal currents.

Exxon is preparing to build a guyed tower for installation in the Gulf of Mexico at a water depth of over 300 m.

Artificial Islands

A further permanent exploration platform is the artificial island of the type constructed in the shallow waters of the Beaufort Sea (20). They are

constructed from gravel reinforced with sandbags for slope protection. As these units are not structures in the conventional sense and do not pertain to the main theme of this report, they will not be discussed further.

Cold Ocean Units

Other variations and designs of production units have been suggested for the exploitaton of hydrocarbon resources in cold oceans, where, added to the normal hostile environmental conditions, similar to the North Sea, there are the seasonal problems of moving pack-ice and icebergs. Two possible structures suggested as production units for oil and gas in ice conditions are shown in Figs. 1.11 and 1.12, and are further described in (21,22,23). A point to note in these designs is the conical structure at the "water line" designed to break the moving pack-ice in flexure.

Table 1.1

Mobile Drilling Rigs	Fixed Platforms
Jack-up rigs	Fixed steel jackets
Semi-submersibles	Gravity concrete platforms
Drill ships	Gravity steel platforms
	Tension leg platforms
	Guyed tower platforms
	Artificial islands.

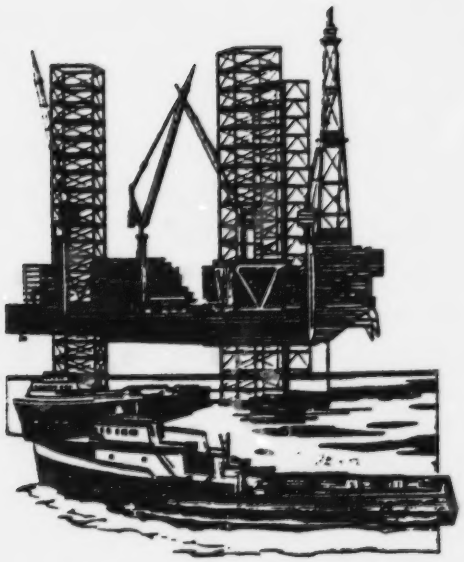


Fig. 1.1. Jack-up drilling unit. (neg 6725)

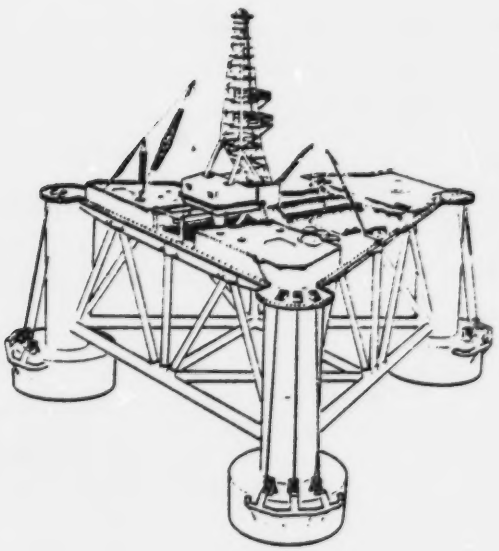


Fig. 1.2. Semisubmersible drilling rig. (neg 6711)

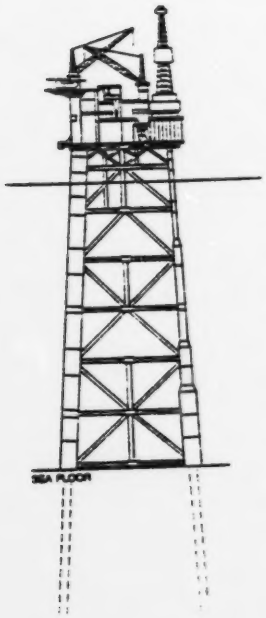


Fig. 1.3. Fixed steel platform. (neg 6686)

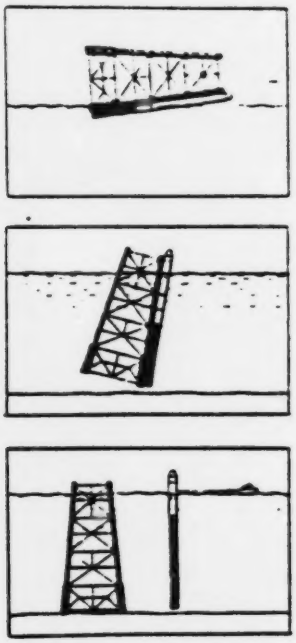


Fig. 1.4. Upending a fixed steel platform. (neg 6720)

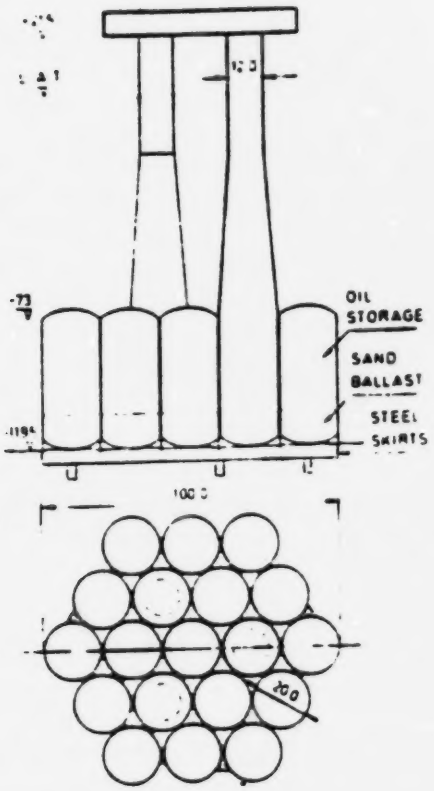


Fig. 1.5. Condeep contrete gravity platform. (neg 6695)

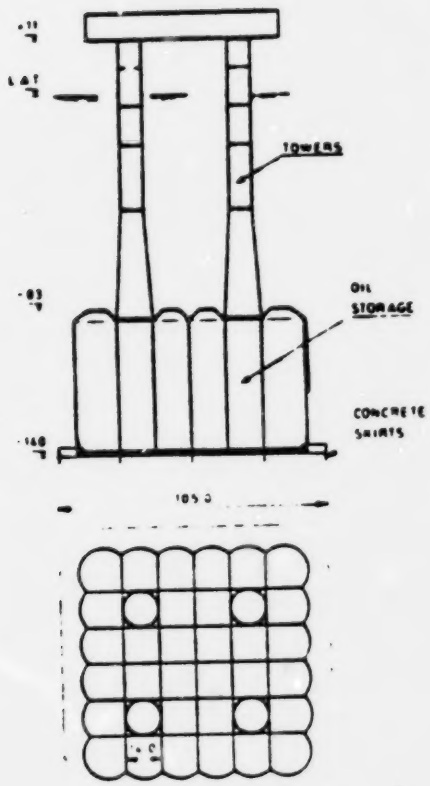


Fig. 1.6. Sea tank concrete gravity platform. (neg 6695)

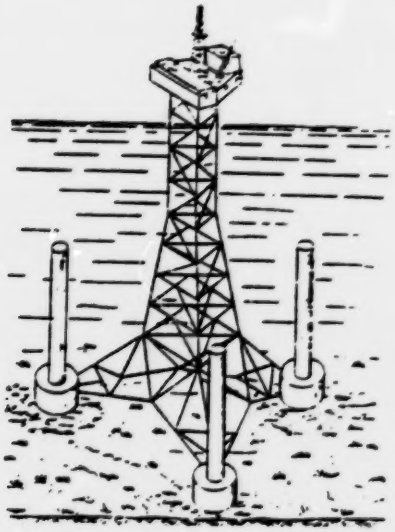


Fig. 1.7. Steel gravity platform. (neg 6724)

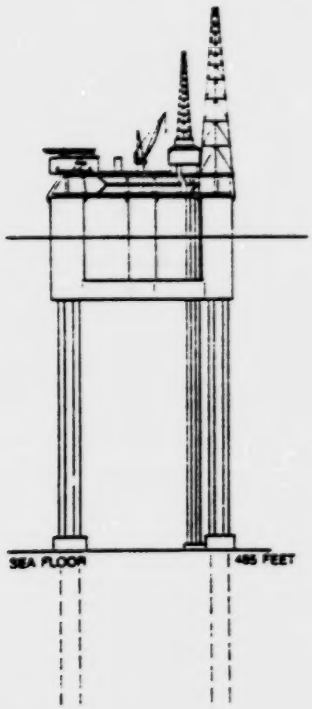


Fig. 1.8. Tension leg platform. (neg 6723)

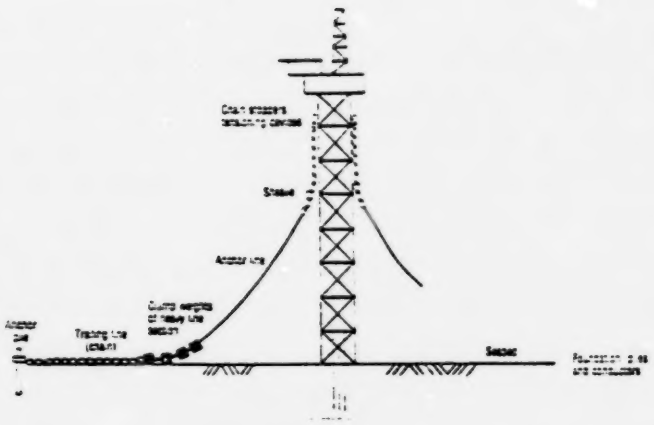


Fig. 1.9. Guyed tower platform. (neg 6710)

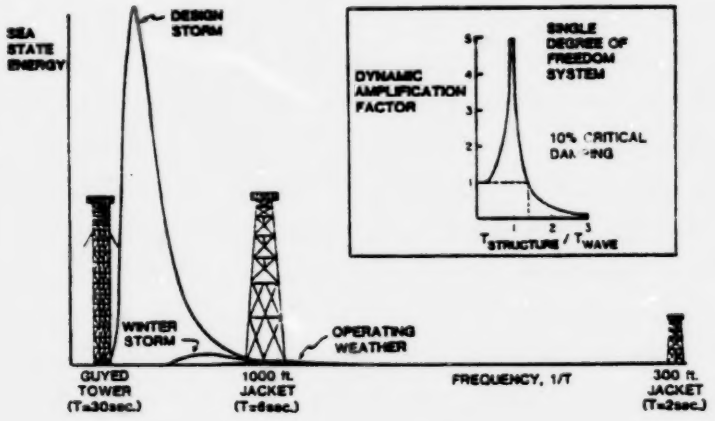


Fig. 1.10. Wave-structure dynamic interaction. (neg 6706)

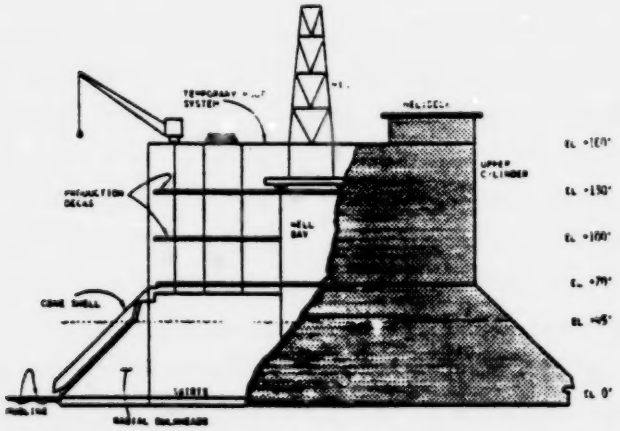


Fig. 1.11. Cossac production platform. (neg 6721)

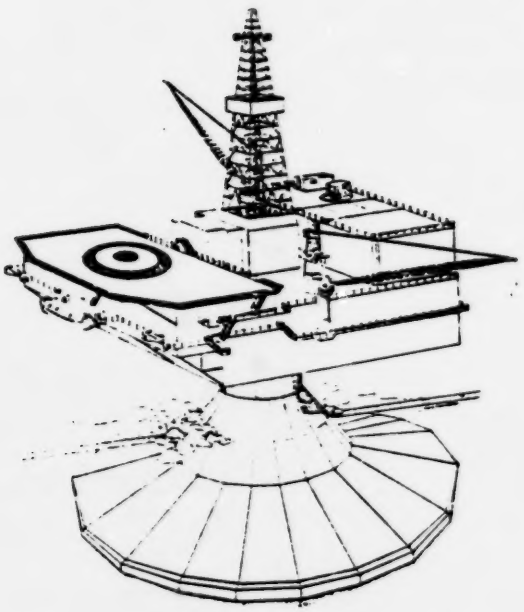


Fig. 1.12. Arctic mobile drilling structure. (neg 6702)

2. DESIGN OF OFFSHORE STRUCTURES

2.1 INTRODUCTION

The great variety of design loads and load combinations and the requirements of adequate safety and serviceability make the design of offshore structures a comprehensive and involved task. The huge dimensions and the grave consequences of possible failure require thorough investigation of the response of these structures to realistic loading and service conditions. These design calculations must be carried out for each phase, including construction, transportation, installation and operational life of the structure.

Various countries have enacted legislation requiring the operators of offshore units to obtain certificates of fitness before the units are permitted to operate in their territorial waters. To assist in this process several organizations have been empowered to certify offshore structures, thus ensuring that all aspects of the design, material selection and construction have been subjected to critical scrutiny by an independent professional body. These organizations and others have published guidelines for the design and construction of offshore units (26-30). Their publications are comprehensive design documents covering general regulations for certification, design principles, construction, inspection and repair procedures and regulations for the selection and design of electrical and mechanical equipment.

Roren and Furness (31) classify the behaviour of offshore structures into three main groups as static, dynamic and enduring behaviour, and list the areas of concern in each of these groups that must be addressed in the design procedure. These design considerations are listed in Table 2.1 and will be referred to as this report progresses.

The methods of analysis of offshore structures have progressed rapidly with the many advances in computational mechanics particularly the finite element method (FEM). The variety of options are summarized in Fig. 2.1 (32).

The first option is to use a static analysis in which the loads are assumed to be quasi-static. This type of analysis is adequate when the period of the structure is less than 2s. At periods greater than 2s inertial forces and dynamic amplification become important and a dynamic analysis is required. This is the case for structures designed to operate in deep waters. In the dynamic case two types of analysis are possible; namely, "frequency domain" in which transient effects are neglected and a steady state solution is obtained,

and "time domain" in which transient and non-linear effects are included using a time-step integration procedure. The frequency domain analysis is usually preferred to save computer time costs, but if effects such as drag, an earthquake or non-linear wave theories are to be studied, a time integration method is required.

The frequency domain analysis may use either a deterministic or probabilistic approach. In the deterministic approach a series of design waves are considered and in the probabilistic approach the seastates and loading are input as spectral curves. Further details of these analyses will be given later in this chapter.

The initial step in the design process is to estimate the loads applied to the structure, Fig. 2.2. This estimate can be a major source of error because of the variability and harshness of the environmental conditions. Waves account for most of the structural loading and produce dynamic effects due to their time dependence, which increase the stresses and cause long term damage, i.e., fatigue.

The seastates that will occur during the life of the structure at the installation site are generally unknown so therefore they must be estimated using wave forecasting techniques. In particular, the design wave of the worst seastate, corresponding to a storm with a 50 or 100 year return period, is required. Two main techniques for defining seastates are represented diagramatically in Fig. 2.3. A short description of these techniques follows: a more detailed discussion may be found in (32,33). The wind distribution or wind rose is transformed into a wave scatter diagram by relating the wind and its duration to the significant wave height and the mean zero crossing period of the wave. This conversion can be carried out using the methods of Darbyshire et al (34). From the wave scatter diagram a wave exceedance diagram or wave spectral density curve may be obtained. The wave exceedance diagram shows the wave frequency as a function of wave height and it can be produced for all waves or waves from a particular direction. In the probabilistic analysis approach, each of the seastates in the wave scatter diagram is transformed into spectra which are applied to the structure and the response is studied in a probabilistic manner.

The next step in the analysis is the complex task of determining the forces exerted on the structure by the waves. The size, i.e. diameter or area which the structural members present to the wave, determines if reflection,

refraction, drag, or inertia are dominant effects in the derivation of the wave forces. A discussion of these effects is given by Hogben (35).

The determination of the forces resulting from any of the above factors can be obtained in two basic steps:

(1) The computation of the kinematic flow field using an appropriate wave theory

(2) The evaluation of the hydrodynamic forces by application of Morison's Equation or diffraction theory (36).

In step one the wave particle motions are determined using solitary wave theory for shallow water waves, Stokes's 5th order wave theory for deep water waves or Airy wave theory for waves of low amplitude. A more complete discussion of the validity of the wave theories is given in (37,33). In step two the inertia and drag forces on slender members are calculated using Morison's equation and in the case of large diameter members using diffraction theory. For a typical steel platform where the member diameters are relatively small, drag and inertia forces are equally important but in a concrete structure with a large diameter column (e.g., 10 m) the inertia forces are dominant.

Diffraction is a very important effect in the determination of the behaviour of large diameter structures. In the determination of wave forces on slender members it is assumed that the presence of the structure has little appreciable effect on the incident wave field; however, for large diameter gravity structures a large profile is presented to the incident wave and the disturbance in the flow field can no longer be neglected. Therefore diffraction wave theory must be used in the calculation of wave forces (38). A lot of work has been reported in the literature on this important topic of wave forces and work still continues (39-42).

Another main area of uncertainty in the design and analysis of offshore structures is in the soil and foundation characteristics and the soil-structure interaction. The main difficulty in this area is the characterization of the soil because of its highly non-linear behaviour. This topic will not be covered in this report but further information may be found in References (43,44).

As the main aim of this report is to comment on the capabilities of FE programs for the analysis of offshore structures, the discussion will be confined to several relevent topics, namely: static analysis, dynamic analysis, analysis of nodes, fatigue analysis and crack propagation. A description of these topics follows.

2.2 STATIC ANALYSIS

The static analysis of offshore structures was developed for, and successfully applied to, structures designed for the Gulf of Mexico. This method utilizes the so-called "design wave" loading technique and it is assumed that if the structure can withstand this extreme static load without failure it will be safe for its service life. The wave forces are derived from environmental conditions forecast at the installation site as outlined in the previous section. These loads are converted to equivalent static loads and combined with estimated deck loads to form static load cases for the analysis. Generally the deflections and stresses are calculated for several wave heights and wave positions and a more detailed stress analysis is only performed for problem areas. These areas are usually joints at critical positions and a more detailed analysis of the nodes will be described in section 2.4. The stresses obtained from the analysis are checked against allowable stresses in the design codes (29,30).

Such simple static analyses give adequate representation of the forces and moments in these structures. Thorough understanding of the elastic behaviour of the structure under static loading is a necessary prerequisite to more elaborate dynamic or fatigue analyses.

2.3 DYNAMIC ANALYSIS

As offshore structures are designed for increasing depths of water the period of the structure increases and at natural periods of 2-4s dynamic amplification becomes important and a dynamic analysis is required. For example in the dynamic case, during the application of the largest design wave the peak deflections and stresses are likely to be 10% greater than those predicted by the static analysis.

In a dynamic analysis the structure is modelled on a three dimensional space frame of beam elements and the analysis is generally carried out using a commercially available FE package.

The dynamic equations of motion for such a system may be written as:

$$[M]\{\ddot{U}\} + [C]\{\dot{U}\} + [K]\{U\} = \{L\} \qquad 2.1$$

where $\{U\}$ $\{U\}$ and $\{U\}$ are the displacement, velocity and acceleration vectors respectively.

[M] is the mass matrix

[C] is the damping matrix

[K] is the stiffness matrix

{L} is the load vector which is a time variant.

The three dimensional solution of the above equation is rarely carried out because of the large number of degrees of freedom a typical structure has. Instead, in many cases, the static solution of the model represented by

$$[K]\{U\} = \{L\} \qquad 2.2$$

is solved and if the dynamic effects are considered significant they are accounted for by dynamic load factors. An example of the use of the dynamic magnification procedure is illustrated by Fig. 2.4 (45). Area A represents a structure with a fundamental period of 2.5s situated in waves with periods between 7 and 11s. The dynamic amplification of this situation is about 1.0. Area B represents a structure with a fundamental period of 5.0s in the same waves, and in this case the dynamic amplification can reach a value of 3.0. It has been shown that the dynamic amplification can vary from place to place in the structure and therefore this method of factoring the static analysis results could lead to either high or low stresses.

Using this dynamic structural model the natural frequencies and mode shapes of the structure may be obtained by the solution of the equation

$$[M]\{\ddot{U}\} + [K]\{U\} = 0 \qquad 2.3$$

An example of mode shapes obtained by such an analysis is shown in Fig. 2.5.

As stated in section 2.1 two distinct approaches to dynamic analyses are possible, i.e. frequency domain and time domain. In both approaches the initial information required with regards to model description is the same as for the static design wave case with the added requirement of structural mass, mass of flooded members, added mass of marine growth and damping, if required.

The deterministic analysis applies a series of design waves to obtain the dynamic response and the subsequent fatigue analysis utilizes the long term wave exceedance diagram. The probabilistic method of analysis recognizes the almost random nature of the seastates by representing them as frequency spectra. Each seastate is represented by a spectrum as shown in Fig. 2.6a,

which is essentially a variance density over the frequency range. This curve is known as the power spectral density curve and represents the water surface elevation components throughout the frequency range. These seastate spectra are generally represented by the JONSWAP or Pierson-Moskowitz analytical form of wave spectra (46,47). The Pierson-Moskowitz repesentation applies to waves generated by constant wind over long fetches for long times and the JONSWAP (Joint North Sea Wave Project) representation applies to fetch-limited wave spectra.

The next step in this analysis is to create stress transfer functions from the results of dynamic response to a number of regular wave trains. These functions represent the stress amplitude per unit wave amplitude at the wave frequency, as shown in Fig. 2.6b. By multiplying the stress transfer function and the wave spectra a stress response spectrum is obtained, 2.6c, for the stress point under consideration. A flow chart of the steps in this analysis is shown in Fig. 2.7 and, as shown, this analysis can be extended to study accumulated fatigue damage. The disadvantages of this analysis as compared to design wave dynamic analysis are cost, computing time and computer capacity required, because one analysis is required for each seastate.

The alternative procedure to frequency domain analysis is time-domain or time-step history analysis. In this approach the structure is subjected to time-step variations of load and the response of the structure is determined by integrating the equations of motion in a step by step manner. It is possible by using this procedure to include non-linearities such as drag and non-linear wave theories in the analysis and to investigate transient effects such as earthquakes. The time history analysis is considered to be exact and yields very accurate results; however, this analysis is involved, generates voluminous data and is extremely demanding of computer capacity and budget. A comparison of the spectrum approach and the time history approach to the analysis of a platform with respect to seismic events is given in (48).

2.4 STRESS ANALYSIS OF JOINTS

The stress analysis of tubular joints, Fig. 2.8 is an important aspect in the overall integrity of offshore structures and as such has received much attention in the literature, (49,50). The ultimate goal in the stress analysis of a tubular joint is to obtain a value of the design stress which is characteristic of the joint and its loading. Until the 1970's the strength of a joint was characterised by the nominal stress in the chord.

This was the punching shear concept and although simplistic, has been useful in correlating test data and formulating design criteria. The punching shear stress, V_p, for a simple joint may be calculated from

$$V_p = \tau \left[\frac{f_a \sin\theta}{k_a} + \frac{f_b}{k_b} \right] \qquad 2.4$$

where f_a is the axial stress and k_a and k_b are obtained from Fig. 2.9. f_b is calculated from

$$f_b = \sqrt{f_{by}^2 + f_{bz}^2} \qquad 2.5$$

where f_{by} and f_{bz} are the in-plane and out-of-plane bending stresses respectively. The terminology and geometric parameters for a simple joint are also given in Fig 2.9 (30).

In this approach if first yield is taken as the criterion of failure the elastic theories seriously underpredict the available static strength of a tubular joint. In fact, tubular joints have tremendous reserve strength beyond first yield. It has been shown that at loads of 2.5 to 8 times the load at first yield the joint fails by pullout failure for tension loads or localized collapse of the chord for compression loads . The sources of excess capacity are (51),

(i) . the difference between elastic and plastic bending strength
(ii) restraint to plastic flow caused by triaxial stresses
(iii) strain hardening
(iv) the redistribution of load that occurs as the joint yields.

Therefore empirical design curves make use of this post yield strength of tubular joints.

At present, because of the main concern about fatigue of tubular joints, the design codes have adopted an approach of characterising the joint by a maximum local stress (27,29,30). This stress can then be used with the various S-N curves corresponding to the type of joint, i.e. American Welding Society X curve or Lloyds Basic curve. The maximum stress or "hot spot" stress at the joint is now more or less well defined and is usually calculated using thin shell theory.

In the future a fracture mechanics approach might be perfected and used for the design of tubular joints. This approach will take into account the effects at the toe of the welds where cracks occur, the initiation period of such cracks and the redistribution of stress in the region of the crack as it propagates. Research is at present taking place in this area but much work remains to be done.

The calculation or measurement of a numerical value of the design stress for a tubular joint can be difficult because of the complex nature of the joint. Therefore the stress analysis involves three stages. First, a global analysis of the structure is performed, modelling the structure as realistically as possible by a network of beams. The loadings from waves, winds etc. are applied to the model as outlined previously. This analysis yields the forces and bending moments in each member of the structure, hence the nominal stress in the straight tubular sections can be obtained. Secondly, the analysis of the joint is then carried out by modelling the intersection on thin shell elements which yield the forces and stresses necessary to maintain compatibility of displacements along the line of intersection. In this analysis the weld is not included in the model and the stress obtained is usually termed the "geometric stress" for the idealized joint geometry. Finally, it is possible to complete a three dimensional analysis in the neighbourhood of the weld taking into account the actual local geometry reproducing the notch effect of the weld which gives rise to local stresses corresponding to the classical definition of stress concentration.

The A.W.S. code (52) defines the design stress as "the total range of worst hot-spot stress or strain on the outside surface of the intersecting members at the toe of the weld joining them - measured after shakedown in model or prototype connection or calculated with the best available theory". This definition is ambiguous and one major positive result of the United Kingdom Offshore Steels Research Program (UKOSRP) and the European Economic Community (EEC) Marine Technology Program is an explicit definition of design stress. This idealized hot-spot stress is defined by linear extrapolation to the toe of the weld of the principal stresses at A and B as shown in Fig. 2.10 (27). This hot-spot stress may be considerably lower than the maximum stress since it does not take into account the rapid increase in stress adjacent to the weld toe. However, this definition of hot-spot stress in consistent with the fatigue S-N design curves in use for tubular joints.

The calculation or measurement of the stress concentration factor for tubular joints may be carried out using several methods eg.

(i) Numerical methods (FEM)

(ii) Acrylic Models

(iii) Photoelasticity

(iv) Measurements on full size steel models

2.4.1 Finite Element Method

Numerical solutions of tubular joints using the FEM not only give a numerical value for the hot-spot stress but also give its precise position and the stress distribution in the vicinity of the joint intersection. In applying this method some of the concerns that must be considered by the user are:

(i) the size and distribution of the element in the mesh to provide adequate accuracy at acceptable cost

(ii) the choice of element to give the required results

(iii) the FE program to use, once again with concern for accuracy, cost and ease of use

(iv) the output capabilities of the program to facilitate graphic presentation and easy interpretation of results.

The FEM has been a very powerful tool for the analysis of tubular joints and in the derivation of parametric formulae used in their design.

2.4.2 Acrylic Models

The method of joint analysis using acrylic models was developed by Lloyd's Register of Shipping and involves the measurement of strains, by strain gauges on small scale acrylic models (53). This method is an alternative to the testing of small scale steel models and it gives results similar to those obtained by thin shell theory. In these tests three dimensional effects that are difficult to model on small specimens are absent. The main advantages of this method are that the manufacture of the joints is inexpensive and rapid and, because of the low modules of elasticity of the acrylic, only light testing equipment is required. However, one difficulty exists in this procedure, namely the susceptibility of the acrylic to creep, but this can be overcome by careful testing procedures (54).

2.4.3 Photoelasticity

Photoelastic model analyses of tubular joints allows verification of the results obtained from calculations by thin shell analysis and measurements on full size steel joints. This method also takes account of the three dimensional effects at the toe of the weld, and no extrapolation to the weld toe is necessary because the principal stresses can be obtained at any point. The deflections measured on photoelastic models are applicable to steel joints if a conversion factor based on the model scale and Young's modulus is used. The results obtained using photoelastic analysis show good agreement with numerically derived results (55,56,57).

2.4.4 Full Size Steel Models

The lack of realistic data on the behaviour of large steel nodes prompted the initiation of large scale test programs by countries involved in North Sea oil exploration. The nucleus of the research program was provided by UKOSRP but other work was carried on in Holland, Italy, France, Germany and Norway. This work was primarily concerned with the fatigue behaviour of steel joints but some tests were performed to determine static strength with particular reference to internally stiffened nodes. The full size steel joints were covered extensively by strain gauges and tested to failure. A great variety of tubular joints were tested under various loading and environmental conditions, Fig. 2.11, and the results are published in (50). A survey of these results is given in (58) and (59). Other results on full scale tests on steel joints were reported at a previous conference (49).

An example of a comparison of the three methods of stress analysis for a steel joint is shown in Fig. 2.12 (60).

2.4.5 Parametric Equations

Perhaps the ultimate goal of stress analysis is the production of parametric equations that will aid the engineer in his quest to design a joint that meets the service requirements. Three main collections of formulae exist for the design of simple tubular joints and are shown in Fig. 2.13. Kuang et al (61), (Exxon) give formulae produced by FEM, Wordsworth and Smedley (53) (Lloyds) present formulae for T, Y and X joints based on acrylic model tests and Gibstein (63) (DNV) published formulae for T joints based on FE results. The area of validity for these three sets of formulae are given in Table 2.2 and a comparison of the values of stress concentration factors for joints

tested in France in the ECSC program are given in Tables 2.3 and 2.4. Unfortunately these formulae only exist for simple geometries and therefore in order to use them for real joints it is necessary to consider the different parts of the joint separately without knowing if such a simplification is justified or indeed safe.

2.5 FATIGUE ANALYSIS OF OFFSHORE STRUCTURES

One of the most important aspects of the endurability of offshore units is the ability of the structure to withstand the time varying stresses caused by the cyclic loading of the waves. The ability of the structure to withstand these fluctuations of stress decreases with the number of cycles which leads to the development and propagation of cracks. The fatigue damage sustained at a point in an offshore structure will depend on the stress history during its lifetime and this, together with the added difficulties of irregular sea conditions, size, dynamic effects and weld complexitites make fatigue life calculations a very complex task. The detailed theory of the various fatigue analysis procedures are covered in references (64) to (68) and therefore only a brief introduction will be included here.

In offshore structures, generally tubular in construction, the intersection of the chord and brace induce high local stresses adjacent to the connecting weld and it is here that fatigue damage will usually occur. Therefore the fatigue analysis is highly dependent on the accuracy with which this high local stress at the hot-spot can be calculated. The design stress range, $\Delta\sigma$ is taken as the idealized hot-spot stress range as defined by Fig. 2.10 for design purposes.

In the recent past, fatigue damage has been discovered on many offshore structures and in some cases fatigue has been identified as the cause of failure of the structure. In December 1965 the "Sea Gem", a jack-up barge, collapsed, capsized and sank in the North Sea. At the time of the accident the barge was being lowered in preparation for moving to another site, when the tie-bars that attached the barge to the supporting legs broke. It was evident from examination of the tie-bars, recovered from the wreck, that fatigue cracks probably initiated brittle failure of these components (69,51).

Fatigue has also caused problems and some failures in mobile offshore structures. In 1966/67 details of major fatigue failure were reported in the SEDCO 135 design of triangular semi-submersible drilling rigs all of which sustained fractures of the aft 2.75 m diameter horizontal braces (51).

In the case of "Sea Quest", also a triangular semi-submersible rig, a fatigue crack 700 mm long was found in the aft horizontal brace after only 89 days of drilling in the North Sea. This failure was attributed to a combination of high stress concentration adjacent to the welds and high cyclic bending stresses. To correct the problem modifications were made to the "Sea Quest" which included an additional horizontal brace betweem the aft caissons (70).

In March 1980 the Alexander L. Kielland, a Pentagone type semi-submersible rig operating in the Norwegian sector of the North Sea, capsized and turned upside down after one of the five vertical columns broke off. The subsequent investigation (8) found that a break occured in one horizontal bracing which resulted in the overloading and failure of the remaining five bracings that connected the vertical column to the rest of the structure. The original failure that initiated the accident was a fatigue failure. It originated at a weld around a cut-out in the bracing where a positioning control device had been welded to the structure. The fatigue crack initiated at the fillet weld and propagated around the circumference of the brace.

To cope with the difficulties of fatigue analysis in offshore structures, several procedures have been developed. To apply these procedures the following information is required:

(i) the service life of the structure
(ii) the predicted environmental conditions for that period at the installation site
(iii) the structural model and joint-stress information
(iv) a fatigue damage model.

The environmental conditions are determined using the methods outlined in section 2.1 and are represented as a series of design waves or as spectral curves for each seastate.

The wave loading on each member are determined using Morison's equation or diffraction theory.

The structural model is generally a three dimensional space frame of beam elements and the analysis is carried out using commercially available FE computer programs. The analysis may be a full three dimensional dynamic analysis or a much simpler static analysis. The static solution has a greatly reduced solution time and dynamic effects may be accounted for by dynamic load factors. The stress concentration factors at the hot-spot are usually calcu-

lated using parametric formulae but in some complex cases a detailed FE analysis of the joint or a full scale test may be required.

The fatigue damage model is based on the Palmgren (71) and Miner (72) method in which the cumulative fatigue damage is obtained by

$$D = \sum_i \frac{n_i}{N_i} \qquad 2.6$$

where D = cumulative damage ratio

i = stress range index

n_i = number of cycles occuring at the i^{th} range

N_i = number of cycles needed for failure at stress range i.

For tubular joints, the cumulative damage calculations are performed at a number of positions around the connection to locate the position with the shortest life. A cumulative damage ratio of unity is usually an acceptable criterion of life, because at each step in the fatigue calculation there is a measure of conservatism and in fixed platforms there is a level of structural redundancy. In the case of mobile structures, where failure of a single member may have catastrophic consequences, much stricter conditions must be imposed and a maximum cumulative damage ratio of less than one may be required by the certifying authority.

The life N_i at a specified stress level is obtained from S-N curves that have been prepared for a number of types of joints using laboratory and full scale tests. Various organizations including the American Welding Society, Lloyd's Register, Welding Institute and the UK Department of Energy have published S-N data for tubular joints. Care should be taken when selecting the appropriate curve to ensure that it represents what is required. For example, S-N design curves published by DOE are based on a mean minus two standard deviations for relevant experimental data, which imply a 98% survival rate, while Gurney and Maddox (74) have published curves based on mean values of test results which imply a 50% survival rate.

Many other factors should also be considered when applying S-N design curves, for example:

(i) Classification of the joint as outlined in the codes.

(ii) Environmental conditions - whether joints are in air, in the splash zone or are protected/unprotected in sea water.

(iii) Post weld heat treatment - if the joint has been stress relieved after welding the allowable fatigue design stress may be increased.

(iv) Weld Improvement - fatigue life may be increased by weld improvement techniques.

(v) Variable Amplitude Loading - joints on offshore structures experience variable amplitude loading, whereas the tests used to produce the S-N design curves are at constant amplitude loading. Research is continuing on the behaviour of tubular joints subjected to variable amplitude loading, but with the limited data available the codes do not as yet recommend any modification to the normal fatigue design procedure.

As with dynamic analyses, the methods of analysis may be deterministic or probabilistic, as shown in Fig. 2.14. In the deterministic approach the wave loading process is represented by a number of discrete waves and it is assumed that the wave position that causes maximum or minimum base shear also causes maximum or minimum stresses in all the members. This assumption is good for most members in the structure but care should be exercised when members near the surface or in the splash zone are being considered. Dynamic effects are included as dynamic load factors and stress concentration factors are calculated using parametric formulae. The relationship between wave height and stress range is plotted and finally the stress damage curve is calculated by plotting stress range against number of waves with specific heights. The number of waves with different heights is obtained from the wave exceedance diagram. This cumulative stress history curve is then compared with the S-N design curves using Miner's Rule.

The probabilistic fatigue analysis follows the approach described in section 2.2 and is represented diagramatically in Figs. 2.6 and 2.14. The seastates and structural response are represented by spectral curves. The different seastates for a period of one year are considered. The number of times a seastate produces peaks that exceed a particular stress level is computed from the distribution of stress peaks and a probability curve is then plotted for each seastate against stress level. This curve is a Rayleigh probability distribution. It is possible to calculate the proportion of cycles in a particular stress range from these probability curves and hence find the total number of cycles at that stress range. This is repeated for all seastates and a cumulative stress history curve for one year is obtained. The results are multipled by a stress concentration factor and compared with S-N curves using Miner's rule. This is repeated for each hot-spot.

Fatigue life prediction is very sensitive to stress range and stress concentration, therefore great care must be taken that the stresses at the joints are not underestimated.

2.6 ANALYSIS OF CRACK PROPAGATION

An alternative to the S-N design curve procedure of estimating the life of a tubular joint is the approach of crack propagation analysis using fracture mechanics. In this approach the relationship between fatigue crack propagation rate da/dN, and the stress intensity range, ΔK, is used to predict the life of the structure. This relationship, known as the Paris Law (80) has the form

$$da/dN = C\Delta K^m \qquad 2.7$$

where da/dN is the crack growth

a = crack length

C and m are material constants

and ΔK is calculated from

$$\Delta K = Y\Delta\sigma\sqrt{\pi a} \qquad 2.8$$

$\Delta\sigma$ is the design stress range and Y is a parameter that depends on the geometry, type of loading, and crack shape.

On integration of the above equation the life N may be obtained as,

$$\int_0^N dN = \frac{1}{\pi^{m/2}\, C\, \Delta\sigma^m} \int_{a_i}^{a_f} \frac{da}{Y^m\, a^{m/2}} \qquad 2.9$$

where a_i and a_f are the initial and final crack lengths respectively.

The application of the analysis to tubular joints presents some difficulty. Experimental investigations of fatigue crack growth at the toe of welds in tubular joints have shown that the crack propagates rapidly along the weld but more slowly in the through-thickness direction.

The crack growth therefore presents a very special three dimensional problem. Various investigators have attempted to address this problem by

treating the crack growth rates of a semi-elliptical crack in the region of the hot-spot as having independent growth rates in two directions (78,79).

Another problem arises in taking account of the period of crack initiation represented by a_i. The value of this parameter greatly affects the predicted life of the component. Some investigators have suggested a value of a_i = 0.01 to 0.03 mm. Work on the application of this method in the prediction of the life of tubular joints continues.

The Paris equation does not account for mean stress effects, stress intensity threshold for fatigue crack growth, nor fracture toughness K_c of the material. Various investigators have published equations to take account of these effects. Forman et al (81) proposed an improved Paris Equation

$$\frac{da}{dN} = C \frac{\Delta K^m}{(1-R) K_c - \Delta K} \qquad 2.10$$

where R is the stress ratio $\frac{\sigma_{min}}{\sigma_{max}}$.

Klesnil and Lukas (82) modified the original Paris equation to include the stress intensity threshold ΔK_{th} as follows:

$$da/dN = C (\Delta K^m - \Delta K_{th}^m) \qquad 2.11$$

Schutz (76) combined the previous equations to account for all these effects and published an equation,

$$\frac{dN}{da} = \frac{C (\Delta K^m - \Delta K_{th}^m)}{(1-R)K_c - \Delta K} \qquad 2.12$$

and claims that the equation fits the experimental data well.

The crack growth method of analysis is unlikely to replace the S-N curve method of life prediction for tubular joints in the near future. It does however, present a method of dealing with a cracked structure and demonstrates the influence of a number of variables on fatigue crack propagation. The S-N curve analysis is only applicable to a sound structure. For further discussion on fatigue crack propagation in welded structures the interested reader is referred to references (76-82).

Table 2.1

Static behaviour	- Static stresses and section forces in major members or parts of the structure, including foundation reactions - local stresses and forces - local and global displacements - stress redistributions and cracking - structural performance as regards safety against progressive collapse due to accidental overloads or mishaps - static load-carrying system to sustain various loads in different design conditions and limit states.
Dynamic behaviour	- Characteristics of dynamically loaded structure - dynamic stresses and displacements - magnification characteristics, i.e. dynamic load factors (DLF) - damping characteristics of structure and foundation system - vibrations in local members - dynamic performance with respect to earthquakes - long-term dynamic stress responses at sections where fatigue is important.
Enduring behaviour	- Change of structural properties during the service life period, including changes of the static/dynamic foundation system - excessive foundation settlements, in particular differential ones - cumulative fatigue stress damage in critical zones during lifetime - fatigue cracks or low-cycle fatigue due to repeated yielding - corrosion of steel members, including concrete reinforcement - durability, including foundation scour - maintenance and repairs - general serviceability and structural fitness during the service life period.

Table 2.2 Validity limits for parametric formulae of various origin.

Parameter	Exxon	DNV		Lloyd's
		Chord	Brace	
α	6.67 - 40		7 - 16	8 - 40
β	0.3 - 0.8	0.225 - 0.9	0.3 - 0.9	0.13 - 1.0
γ	8.33 - 33.3	10 - 30	10 - 30	12 - 32
τ	0.2 - 0.8	0.4 - 1.0	0.47 - 1.0	0.25 - 1.0
θ	0 - (π/2)	π/2	π/2	(π/6) - (π/2)
ζ	0.01 - 1.0			

Table 2.3 SCF in IRSID joints a) calculated and experimental data.

Load	Joint	Real Geometry		FEM calculations SCF		FEM calculations SNCF		SCF measured	
		chord	brace	chord	brace	chord	brace	chord	brace
	A	D 473 T 22.8 L 2000	d 341 t 21.5	11.90	7.62	10.56	6.79	11.0	7.5
	B	D 684 T 40 L 2000	d 342 t 22.4	6.27	4.95	5.33	4.39	4.7	4.4
	C	D 1281 T 77.6 L 2000	d 343 t 22.4	1.91	1.76	1.49	1.58	1.6	1.7
	D	D 949 T 41.6 L 2000	d 682 t 41.4	11.13	6.78	9.90	6.01	10.0	7.0
	E	D 1280 T 75 L 2000	d 683 t 40	3.98	3.36	3.37	2.98	3.8	3.2
	A'	D 472 T 22.3 L 2000	d 341 t 22	3.33	1.94	2.75	1.66	3.04	1.84
	B'	D 685 T 40 L 2000	d 343 t 22	1.89	1.67	1.47	1.45	1.45	1.45
	C'	D 1275 T 75 L 2000	d 343 t 22.8	0.97	1.30	0.95	1.18	0.62	1.33
	D'	D 947 T 44 L 2000	d 683 t 43.5	3.91	1.92	3.17	1.52	2.72	-
	E'	D 1273 T 76.7 L 2000	d 684 t 43.5	1.98	1.87	1.51	1.55	1.56	1.71

Table 2.4 SCF in IRSID joints b) values obtained from parametric formulae.

Joint	Exxon: T		DNV: T		Lloyd's: T		Lloyd's: X	
	chord	brace	chord	brace	chord	brace	chord	brace
A	8.7	10.0	9.6	6.8	9.4	6.9	15.7	10.9
B	4.9	7.1	4.7	4.6	5.4	4.4	7.8	5.9
C	2.2	3.9	1.6	2.6	2.2	2.4	2.5	2.6
D	9.8	10.3	10.9	6.9	10.9	7.9	18.3	12.6
E	4.3	6.1	4.1	4.1	5.0	4.2	7.6	5.8
A'	2.9	2.5	3.8	2.2	3.4	3.1	3.4	3.1
B'	1.6	2.2	2.0	1.8	2.0	2.2	2.0	2.2
C'	0.9	2.2	1.0	1.5	1.1	1.7	1.1	1.7
D'	2.9	2.5	3.8	2.2	3.4	3.2	3.4	3.2
E'	1.6	2.2	2.0	1.8	2.0	2.3	2.0	2.3

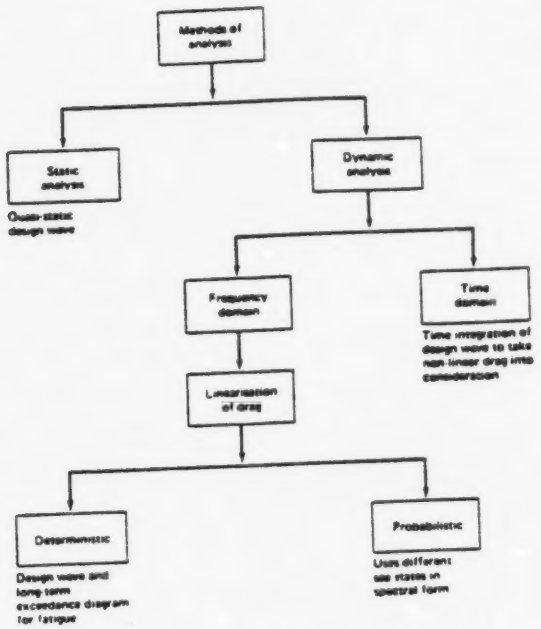


Fig. 2.1. Methods of analysis. (neg 6726)

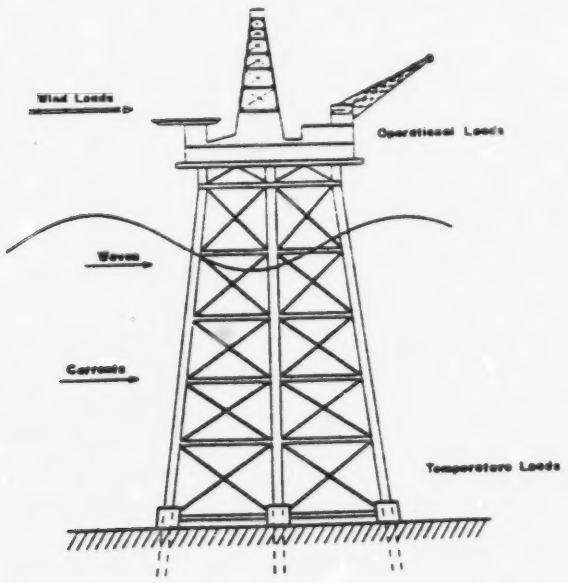


Fig. 2.2. Loads acting on Offshore Structures. (neg 6727)

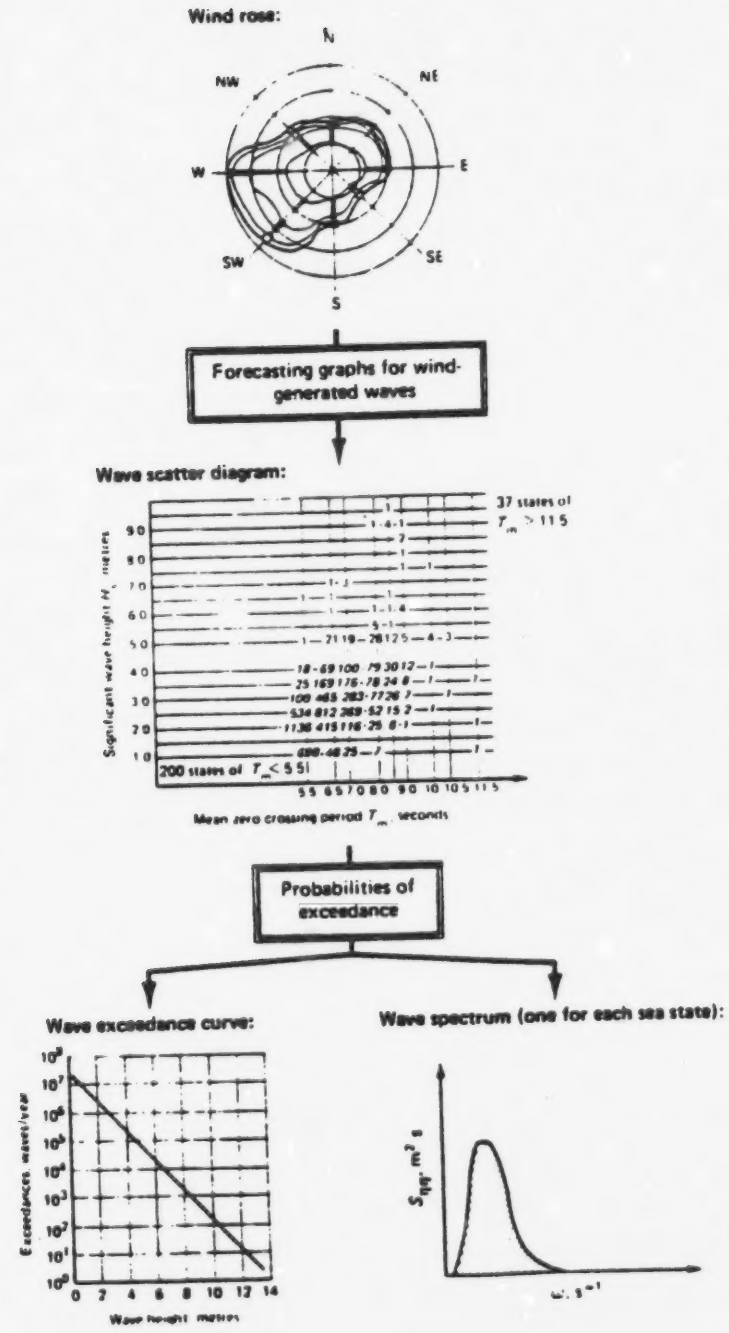


Fig. 2.3. Two main techniques used for defining seastates. (neg 6687)

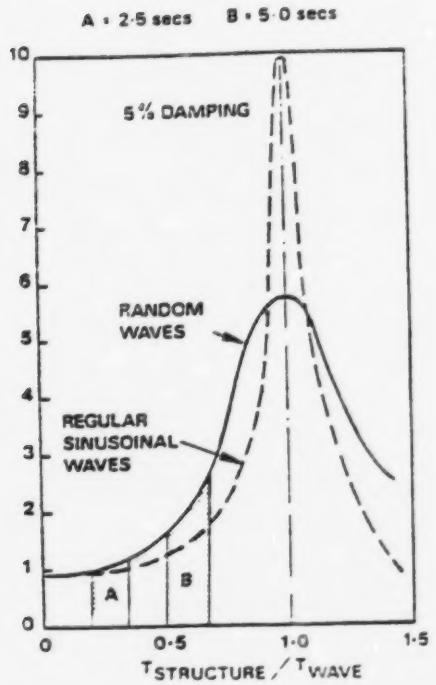


Fig. 2.4. Dynamic amplification factor (forces deflections) (neg 6728)

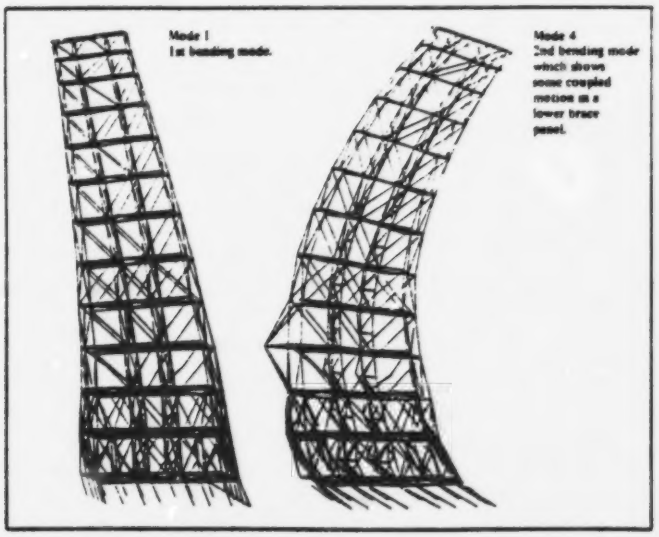


Fig. 2.5 Two bending modes for a jacket designed for 250m water depth.

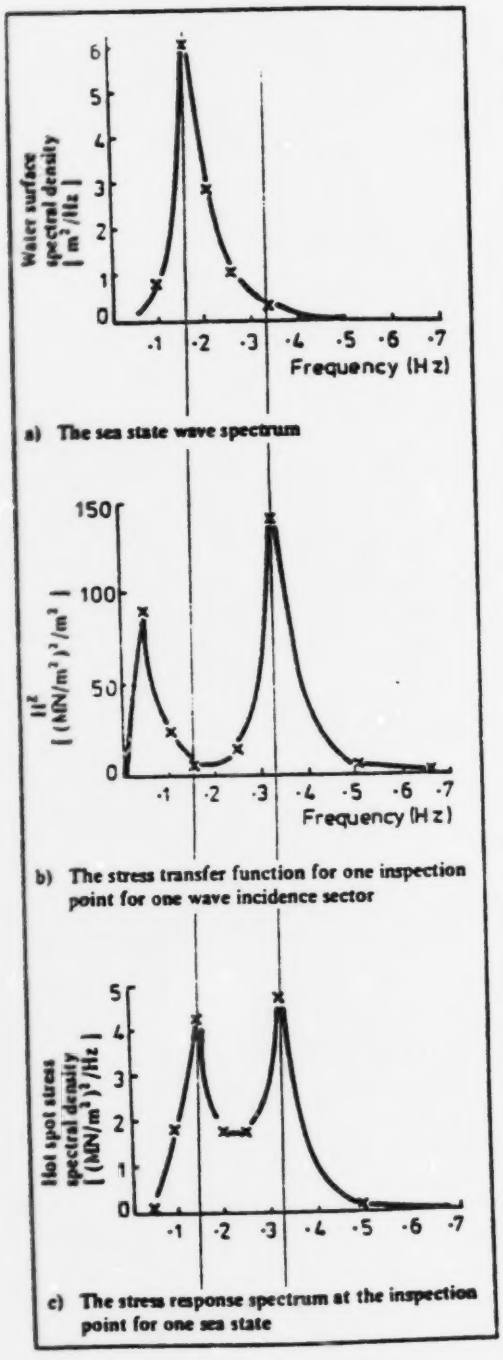

Water surface spectral density [m²/Hz]
Frequency (Hz)
a) The sea state wave spectrum
|H|² [(MN/m²)²/m²]
Frequency (Hz)
b) The stress transfer function for one inspection point for one wave incidence sector
Hot spot stress spectral density [(MN/m²)²/Hz]
c) The stress response spectrum at the inspection point for one sea state

Fig. 2.6. Stress response under the action of a partiuclar sea state. (neg 6694)

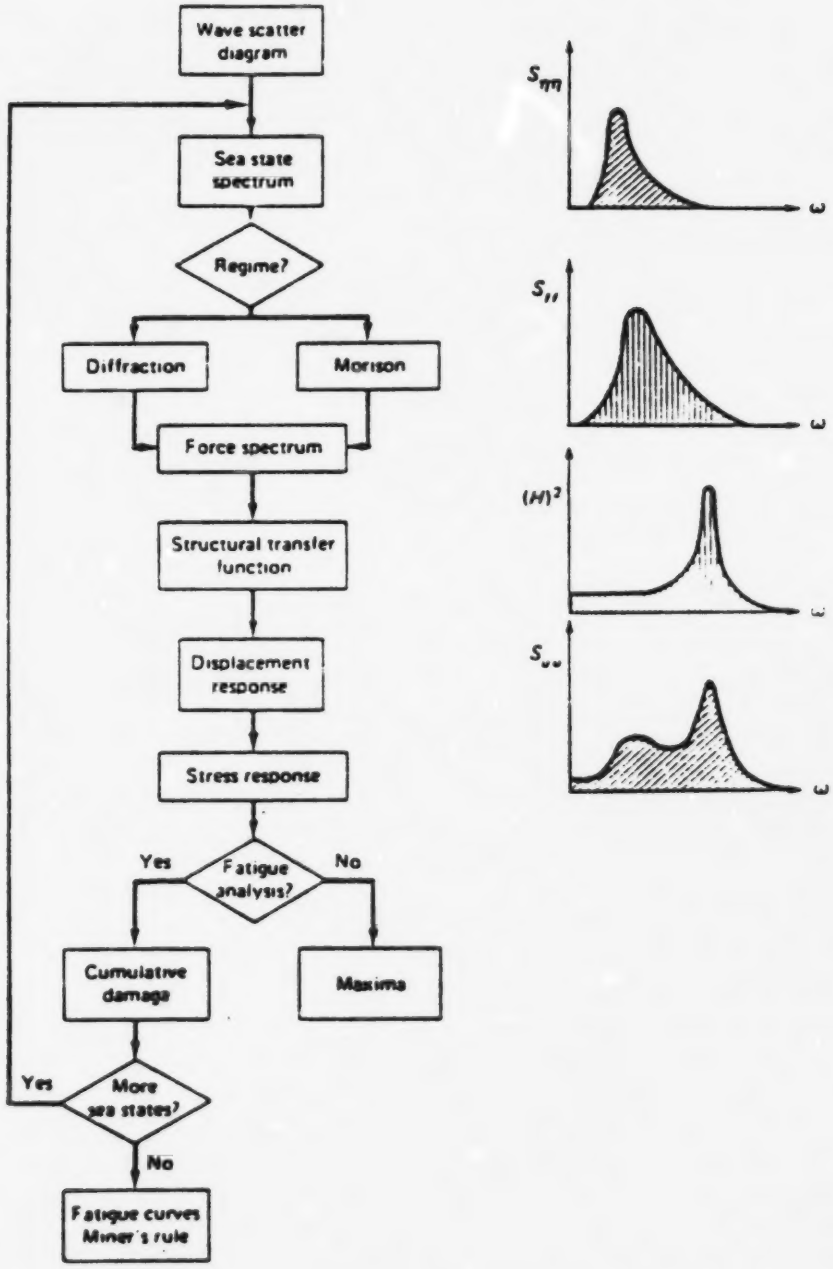


Fig. 2.7. Probabilistic analysis. (neg 6708)

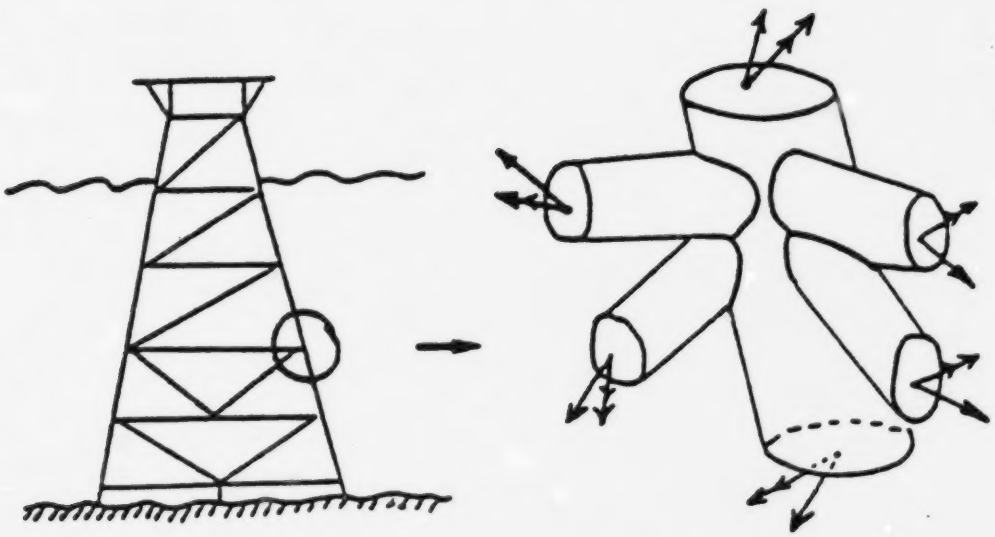
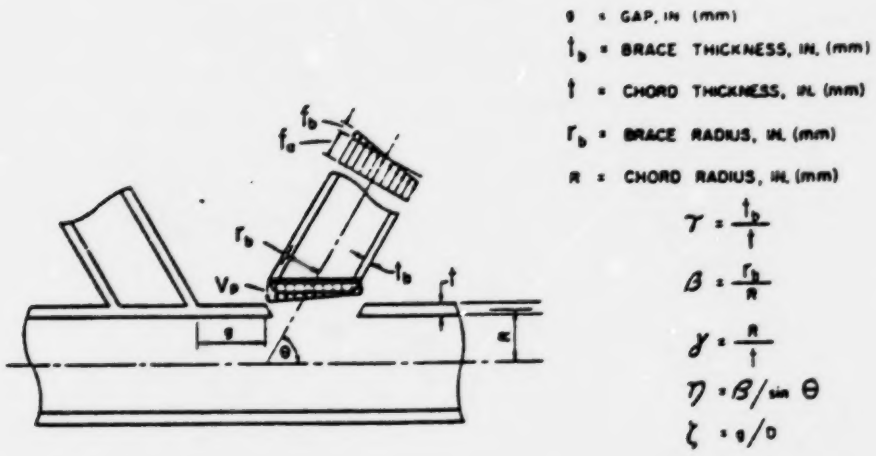
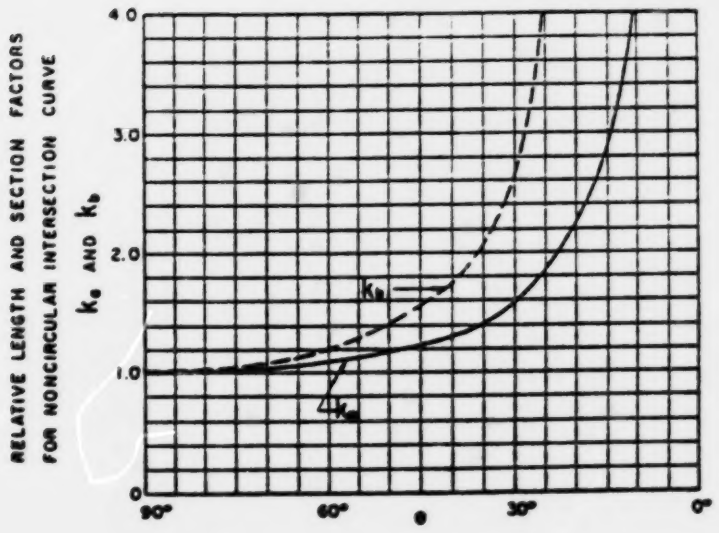


Fig. 2.8. Tubular joint. (neg 6722)



TERMINOLOGY AND GEOMETRIC PARAMETERS
FOR SIMPLE TUBULAR CONNECTIONS



RELATIVE LENGTH AND SECTION FACTORS
FOR NONCIRCULAR INTERSECTION CURVE

Fig. 2.9. (neg 6697)

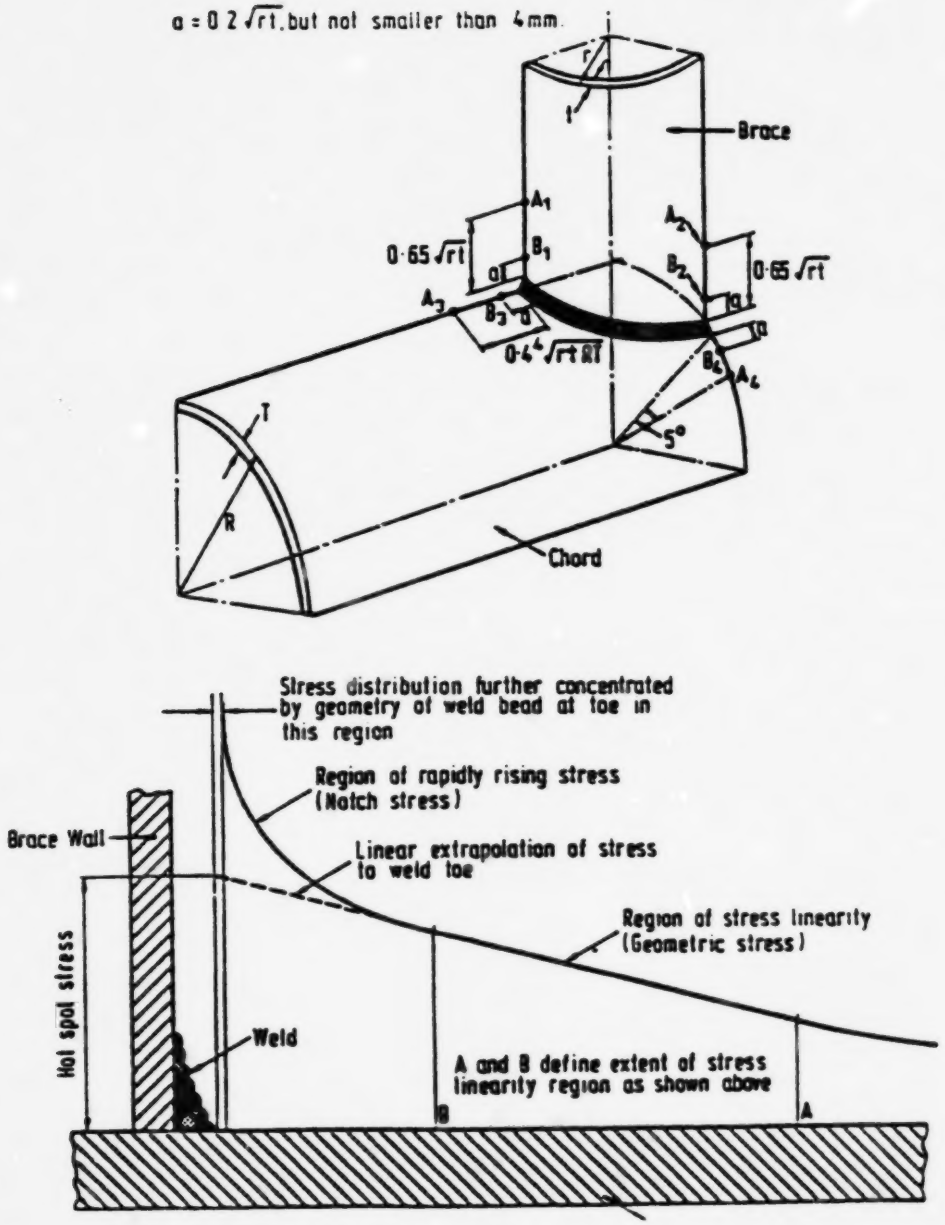

a = 0·2 √rt, but not smaller than 4mm
Brace
Chord
0·65 √rt
0·4 √rt RT
5°
Stress distribution further concentrated by geometry of weld bead at toe in this region
Region of rapidly rising stress (Notch stress)
Brace Wall
Linear extrapolation of stress to weld toe
Region of stress linearity (Geometric stress)
Hot spot stress
Weld
A and B define extent of stress linearity region as shown above

Fig. 2.10. Definition of idealized hot-spot stress. (neg 6696)

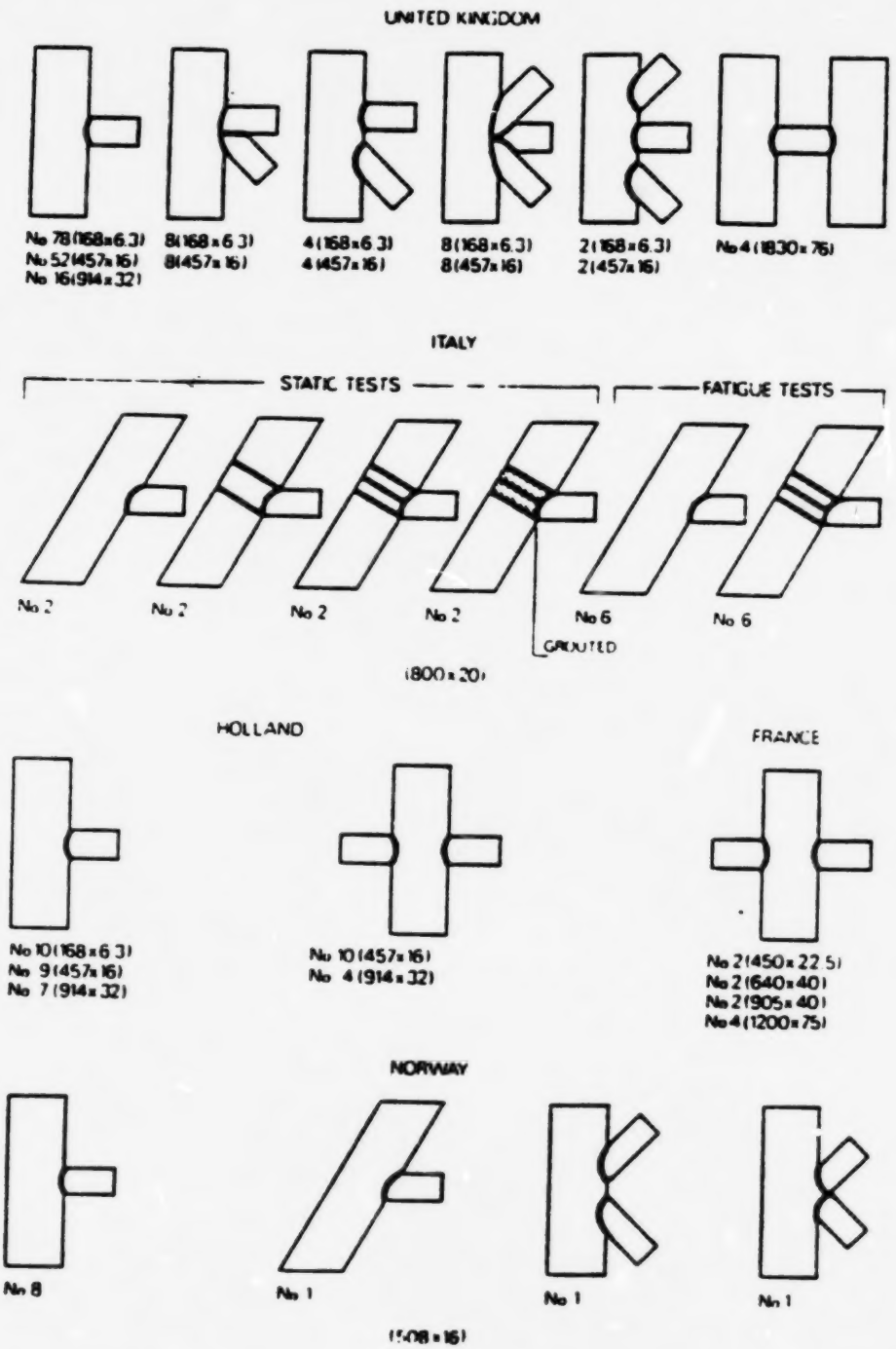


Fig. 2.11. Node Tested - ECSC program. (neg 6690)

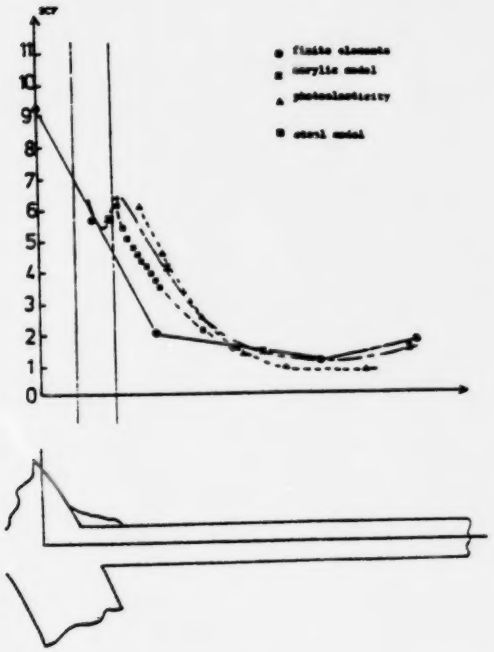


Fig. 2.12. Comparison of methods of analysis joint T1 (UKOSRP). (neg 6718)

Geometry	Source	SCF in the chord	SCF in the brace
T Joint Axial Loading	Exxon 1977	$1.981\ \alpha^{0.557}\alpha^{-1.28}\beta^{1.9}\gamma^{0.866}\tau^{1.111}\sin^{1.684}\theta$	$3.751\ \alpha^{0.12}\ e^{-1.11\beta^2}\ \tau\ \sin^{1.84}\theta\ \gamma^{0.88}$
	DnV 1978	$[1.5-3.88\ (\beta-0.47)^2]\tau^{0.97}\tau^{1.57}\ (\alpha/2)^{0.66}$	$[1.09-1.93\ (\beta-0.5)^2]\gamma^{0.76}\tau^{0.57}(\alpha/2)^{0.12}$
	Lloyd's 1978	$\beta\gamma\tau\ (0.78-6.42\beta^{0.5})\sin^{(1.7\alpha\ 0.7\beta}\theta$	$1.+ 0.63 \times$ SCF in the chord
Double T Joint Axial Loading	Lloyd's 1978	$1.7\beta\gamma\tau\ (2.42-2.28\beta^{2.2})\ \sin^{\beta^2\ (1.5-1.0\cdot\alpha\beta)}\theta$	idem τ
T Joint In-Plane Bending	Exxon 1977	$0.702\beta^{-0.04}\gamma^{0.6}\tau^{0.84}\sin^{0.47}\theta$	$1.301\ \beta^{-0.16}\gamma^{0.21}\tau^{0.80}\sin^{0.81}\theta$
	DnV 1978	$[1.65-1.1(\beta-0.42)^2]\ \gamma^{0.16}\tau^{1.05}$	$[.95-0.65(\beta-0.41)^2]\gamma^{0.15}\tau^{0.25}$
	Lloyd's 1978	$0.75\gamma^{0.6}\tau^{0.8}\ (1.6\beta^{0.25}-0.7\beta^2)\sin^{(1.5-1.4\beta}\theta$	$1 + 0.63 \times$ SCF in the chord
Double T Joint In-Plane Bending	Lloyd's 1978	idem τ	idem τ
T Joint Out-of-Plane Bending	Exxon 1977	$\begin{cases}(\beta<0.55): 1.024\beta^{0.787}\gamma^{1.014}\tau^{0.688}\sin^{1.557}\theta \\ (0.55<\beta<0.75): 0.462\beta^{-0.615}\gamma^{1.014}\tau^{0.688}\sin^{1.557}\theta\end{cases}$	$1.522\beta^{0.881}\gamma^{0.852}\tau^{0.543}\sin^{2.033}\theta$ $0.796\ \beta^{-0.281}\gamma^{0.852}\tau^{0.543}\sin^{2.033}\theta$
	DnV 1978	$[1.01-3.36(\beta-0.64)^2]\gamma^{0.98}\tau^{1.16}$	$(0.76-1.92\ (\beta-0.72)^2]\ \gamma^{0.09}\tau^{0.47}$
	Lloyd's 1978	$\beta\gamma\tau\ (1.6-1.15\beta^5)\ \sin^{(1.35\ \beta^2}\theta$	$1 + 0.63 \times$ SCF in the chord
Double T Joint Out-of-Plane Bending	Lloyd's 1978	$\beta\gamma\tau\ (1.56-1.46\beta^5)\ \sin^{\beta^2(1.5-1.0\cdot\alpha\beta)}\theta$	idem τ
K Joint Axial Loading	Exxon 1977	$1.500\ \beta^{-0.059}\gamma^{0.666}\tau^{1.106}\zeta^{0.067}\sin^{1.521}\theta$	$0.920\ \beta^{-0.441}\gamma^{0.157}\tau^{0.068}\zeta^{0.056}\ e^{1.44\theta}\sin\theta$
K Joint In-Plane Bending	Exxon 1977	$1.822\ \beta^{0.06}\gamma^{0.48}\tau^{0.94}\sin^{0.9}\theta$	$2.827\ \beta^{-0.35}\tau^{0.35}\sin^{0.5}\theta$

Fig. 2.13. Parametric formulae for simple joints. (neg 6719)

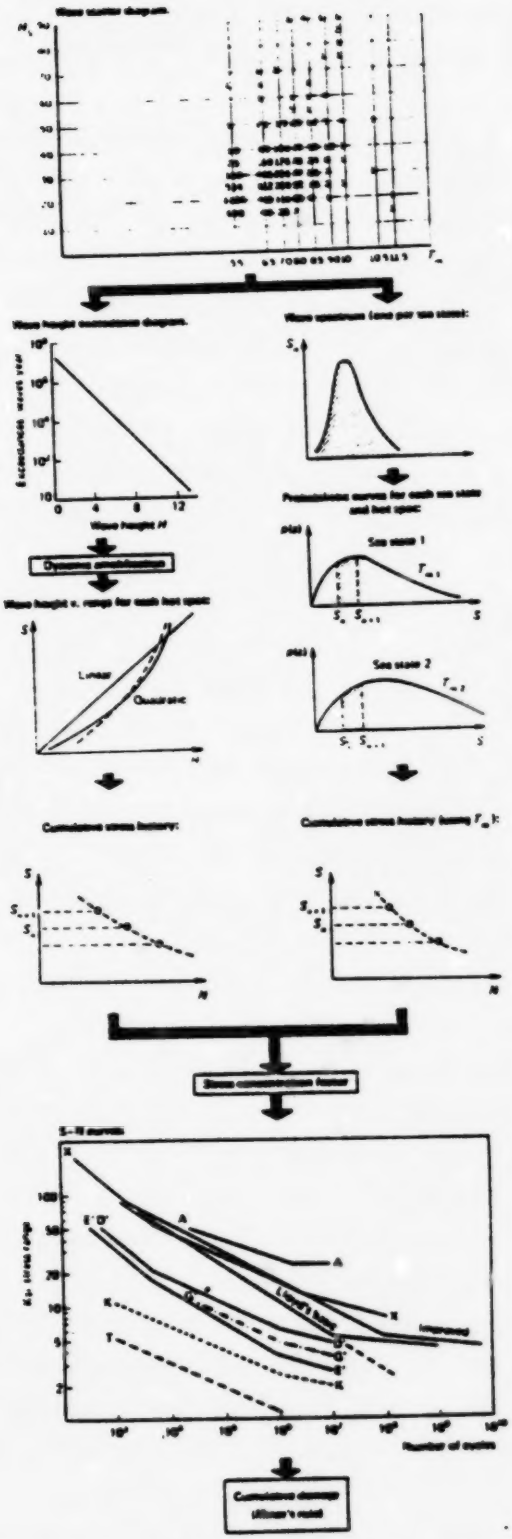


Fig. 2.14. Methods of analysis.

3. GENERAL PURPOSE FINITE ELEMENT PROGRAMS FOR THE ANALYSIS OF OFFSHORE STRUCTURES

This chapter gives a brief introduction to the FEM giving the basic matrix equations and describing some of the analyses that may be undertaken using this method. A general description of the capabilities of general purpose finite element programs is also given. Finally the FE computer systems that have offshore structure analysis capabilities are described.

The information collected on these systems was obtained from organizations that developed the programs, promotional literature, computing bureaux and private communications. The degree of detail of the information therefore depends on the amount of information published or disclosed by the organization concerned. Further details on the computer packages may be obtained by contacting the organizations directly. A list of addresses is given in Appendix I for this reason.

3.1 GENERAL REVIEW OF FINITE ELEMENT METHOD (FEM)

The basic finite element theory is well documented (83)(84) and the following work reviews the basic matrix equations that arise in the analysis procedures of general purpose finite element programs used in the analysis of offshore structures. In essence the finite element method represents the structure of interest by a number of structural elements such that each element covers a relatively small piece of the structure. Within each element assumptions are made on the variation of displacement or force, which define either the displacement method or force method of analysis. The displacement method has been the most common in finite element programs.

The displacements are assumed to vary across each element in a manner described by shape functions that are weighted in terms of the displacements at the nodes on the boundary or within the element. The displacements at the nodes are written as the vector {U}, which comprises the components of displacement or their spatial derivatives.

At any point in the element the strain vector can be separated into primary strains $\{\varepsilon\}_e$ due to mechanical loads and secondary strains $\{\varepsilon\}_0$ due to thermal or residual effects. In linear elastic behaviour the stress vector may be represented by

$$\{\sigma\} = [D] (\{\varepsilon\}_e - \{\varepsilon\}_0) \qquad 3.1$$

where [D] is a square matrix of elastic constants.

The total strain vector is related to the nodal displacement vector by,

$$\{\varepsilon\} = [B]\{U\}^{\ell} \tag{3.2}$$

where [B] is a matrix of the derivatives of the assumed shape functions for the element. It is now possible to write the strain energy density at a point as,

$$\Omega = 1/2\ \{\sigma\}^{T} \left\{\{\varepsilon\} - \{\varepsilon_o\}\right\} \tag{3.3}$$

and to obtain the strain energy of the element as

$$\begin{aligned}\Omega^{\ell} = 1/2\ \{u\}^{\ell T}\Bigg(\int_{\ell} [B]^{T}\,[D]\,[B]\,dV\Bigg) \\ - \{u\}^{\ell T}\int_{\ell} [B]^{T}\,[D]\,\{\varepsilon_o\}\,dV \\ + 1/2\int_{\ell} \{\varepsilon_o\}^{T}\,[D]\,\{\varepsilon_o\}\,dV\end{aligned} \tag{3.4}$$

The element stiffness is defined as,

$$[K]^{\ell} = \int_{\ell} [B]^{T}\,[D]\,[B]\,dV \tag{3.5}$$

and the potential energy of the element as

$$P^{\ell} = \Omega^{\ell} - \{U\}^{\ell^{T}}\,\{L\}_{e}^{\ell} \tag{3.6}$$

where $[L]_{e}^{\ell}$ is the mechanical load vector which gives rise to primary strains only. The components of $[L]_{e}^{\ell}$ correspond term by term with displacements $\{U\}^{\ell}$ for the element. If a secondary load vector is introduced as

$$\{L\}_{o}^{\ell} = \int_{\ell} [B]^{T}\,[D]\,\{\varepsilon_o\}\,dV \tag{3.7}$$

and

$$W^{\ell} = 1/2\int_{\ell}\{\epsilon_o\}^T[B]\ \{\epsilon\}_o\ dV \qquad 3.8$$

then the potential energy of the element may be written as,

$$P^{\ell} = 1/2\ \{U\}^{\ell^T}[K]^{\ell}\ \{U\}^{\ell} - \{U\}^{\ell^T}\ \{L\}_0^{\ell} + W^{\ell} - \{U\}^{\ell^T}\ \{L\}_e^{\ell} \qquad 3.9$$

The matrices and vectors for every element can now be obtained by superposition, taking into account the distinction between the different nodal displacement or nodal degrees of freedom to form the global equations for the complex structure. Thus, in the complete structure the potential energy is

$$P = 1/2\{U\}^T[K]\{U\} - \{U\}^T\{L\}_T + W \qquad 3.10$$

where [K] is the stiffness matrix of the complete structure, {U} is the displacement vector, and

$$\{L\}_T = \{L\}_e + \{L\}_0 \qquad 3.11$$

is the total load vector comprising the mechanical load vector $\{L\}_e$ and a secondary load vector $\{L\}_0$.

Minimizing the potential energy gives the equilibrium equations,

$$[K]\{U\} = \{L\}_T \qquad 3.12$$

These are the basic equations, (3.12) to be solved in the finite element method. In the linear elastic case a set of linear simultaneous equations arise that can be solved by procedures such as Gaussian elimination. The components of {U} are usually unknown, although it is possible to prescribe any number of them to satisfy the boundary conditions of the structure.

For a given analysis the more elements (and therefore degrees of freedom) used the more accurate the results. A variety of one dimensional (1-D), 2-D and 3-D elements exist in which the displacement is usually in polynomial form. Linear, quadratic, cubic and higher orders of displacement variation are commonly in use and for a given element type the higher the order of polynomial, the fewer the number of elements required to model the structure and achieve the desired accuracy of results. Therefore, the analyst has to compromise on the type and quantity of elements to use in order to ensure sufficient numerical accuracy without incurring excessive computer cost and core storage.

Equation 3.12 is the set of equations governing the static behaviour of the structure, but it is also possible to model the dynamic behaviour using FEM. The dynamic behaviour may be modelled using dynamic equations of motion written as follows,

$$[M]\{\ddot{U}\} + [C]\{\dot{U}\} + [K]\{U\} = \{L\} \qquad 3.13$$

where $\{U\}$ is the displacement vector

$[M]$ is the mass matrix

$[C]$ is the damping matrix

$[K]$ is the stiffness matrix

$\{L\}$ is the load vector, it varies with time.

As stated previously, because of the large number of degrees of freedom in a full 3-D structure the solution of equation 3.13 may involve significant computing resources.

If the matrix $[C]$ is zero, i.e. no damping exists and no forcing vector $\{L\}$ is present, the general equation 3.13 reduces to

$$[M]\{\ddot{U}\} + [K]\{U\} = 0 \qquad 3.14$$

A solution in the simple, real periodic form

$$\{U\} = \{U\}_0 \cos \omega t \qquad 3.15$$

exists if

$$([K] - \omega^2[M])\{U_0\} = 0 \qquad 3.16$$

This is only possible for certain values of ω for which the determinant of the matrix in parentheses is zero. As the determinant is of nth order, i.e. the structure has n degrees of freedom and in general n real roots ω^2 exist. The roots ω^2 give the natural frequencies of the structure. These frequencies (ω_1, ω_2, ω_3, ... ω_n) represent the frequencies of the n nodes of vibration that are possible in the structure.

The FEM may also be used to predict structural response to earthquakes. In this case the definition of an appropriate ground motion history is the most difficult and uncertain phase of the problem. Once a suitable support-excitation history has been established, the calculation of deflections and stresses in any given structure is a standard problem of structural dynamics. The only special feature of the earthquake problem, compared with any other form of dynamic loading, is that the excitation is applied in the form of support motions rather than by external loads. These ground motions are usually expressed in terms of three translational acceleration components.

In this case the equation of motion may be written as,

$$[M]\{\ddot{U}\}^t + [C]\{\dot{U}\} + [K]\{U\} = 0 \qquad 3.17$$

where the superscript t denotes total displacement. When it is noted that

$$\{U\}^t = \{U\} + \{1\}\, U_g \qquad 3.18$$

where $\{U\}$ is the relative displacement, $\{1\}$ is a column of ones and U_g is the ground displacement. Therefore, equation 3.17 may be written in terms of relative displacement

$$[M]\{\ddot{U}\} + [C]\{\dot{U}\} + [K]\{U\} = \{L(t)\}_{eff} \qquad 3.19$$

where

$$\{L(t)\}_{eff} = -[M]\{1\}\ddot{U}_g(t) \qquad 3.20$$

Equation 3.19 may be solved directly by numerical integration of the coupled equation, however, the earthquake analysis of linear structures may be ade-

quately performed by transformation to modal coordinates because the ground motions tend to strongly excite only the lowest modes of vibration. A more detailed description of this procedure is given in (85).

3.2 GENERAL PURPOSE FINITE ELEMENT PROGRAMS

The finite element method is a powerful tool for analyzing physical problems and its development has been very rapid during the last decade. During that time a large number of finite element computer programs have been developed. Many of these programs have been special purpose with limited applicability, but some have been large scale general purpose programs that are available on the commercial market. Typically these programs may consist of 100,000 to 300,000 Fortran statements and 1,000 to 3,000 subroutines and may have taken 100-150 man-years to develop. Recent developments of general purpose finite element programs have been to make them user oriented (i.e., easy to use), to increase the problem size and to increase the efficiency with which solutions may be obtained.

Some of the factors which may influence the selection of general purpose finite element programs from the "ease-of-use" point of view are listed below:

(i) Analysis Capabilities

These include the range of applications and limitations of the computer code. The limitations include the element library available in the program and the numerical solution procedures adopted in the code.

(ii) User-oriented Features

These features include automatic or semi-automatic mesh generation for data input, data error checking, graphic displays of the original model and presentation of intermediate and final results.

(iii) User Support Facilities

To use a general purpose FE program it is essential that the user manuals, training manuals, programming manuals, sample problems, and test cases are well documented, clear and concise. Other user services are also desirable such as training courses, user meetings and assistance by consultants from the program owners or operators' organization.

A detailed survey of the capabilities of general purpose finite element programs will not be included here. An example of the capabilities of a few of the larger, comprehensive programs that have a rich variety of element types is shown in Table 3.8. For a more complete survey covering almost every com-

puter program on the market the reader is referred to the catalogue by Fredriksson and Mackerle (86). A survey of non-linear finite element programs is given by Noor (87).

The remainder of this chapter will concentrate on general purpose finite element programs with capabilities that are applicable to the analysis and design of offshore structures.

3.3 COMPUTER PACKAGES FOR THE ANALYSIS OF OFFSHORE STRUCTURES

3.3.1 Det Norske Veritas

The Norwegian ship classificaton society, Det norske Veritas, has developed a comprehensive suite of finite element programs known as SESAM-69 (Super Element Structural Analysis program Modules) designed for the analysis of offshore structures. As the name implies the system is of modular construction and the individual programs may be used independently or as part of the larger system. This system was one of the first to use the superelement or multilevel substructuring technique. Using the superelement technique several basic finite elements are combined to form so-called superelements which may be of arbitrary shape and size. This superelement is referred to as the first level, in contrast to the higher level superelements that are built up of superelements from lower levels. The superelements are treated by the program in the same way as the basic finite elements. This procedure has the advantage of permitting the solution of very large and complex structural problems in an economical way. A more detailed description of superelement or multilevel substructuring techniques may be found in (88).

The SESAM-69 system can handle linear elastostatic orthotropic structures, eigenvalue and linear dynamic response analysis and can also be used for elastic-plastic materials and some limited large displacement analyses. The element library in SESAM-69 is rather limited for a general purpose program. A list of these element types and a brief discription of each is given in Table 3.1. Notable capabilities not included in Table 3.1 are the finite elements for linear and non-linear fracture mechanics studies in two and three dimensions.

SESAM-69 has a pre-processor capability for data input generation in two and three dimensions and a graphic display system for model checking. The program also has a post-processor plotting capability for display of stress-contour plots, undeformed/deformed model geometrics and natural vibration mode

shapes. A more detailed description of the SESAM-69 system and its component programs is given in (89-91).

More specifically the DNV/SESAM-69 system has an extensive capability to carry out the analysis of offshore structures. The programs to carry out the required tasks are listed below.

(i) Wave and Current Loading -

NV426

CONLOA these programs calculate the wave and current forces on a fixed structure using Morison's equation

(ii) Soil and Pile Analysis

Static pile analysis

LLPA these programs calculate the ultimate load capacity and load

PY response curve for a given pile or partially embedded pile. The relationships between the lateral soil-resistance and the lateral displacement Y may also be computed

(iii) Structural Analysis

NV337 three dimensional FE program for static and dynamic analysis of frame structures

FEDA FE program for static and dynamic analysis of frame membrane and shell structures. The dynamic analysis may be carried out either in the frequency or time domain.

NV336 the super-element program that is connected to the other FE computer programs (i.e. 337, etc.)

(iv) Stress Checking

NV151 checking of stress against codes (DNV, AISC)

NV348 calculates punching shear in tubular joints and checks with the allowable punching shear according to API guidelines

(v) Fatigue Analysis

NV432 provides a probabilistic fatigue analysis capability for fixed offshore structure

NV378 this program utilizes the Miner-Palmgren accumulated linear damage criteria to predict the life of a structure. The program can account for the changes in direction of the wind and waves during the life of the structure

(vi) Fracture

Using the fracture mechanics capabilities of SESAM-69 it is possible to evaluate the influence of root defects in welded connections and to

carry out an analysis of the propagation of fatigue cracks. For example the NV344 program computes stress intensity factors and the fatigue propagation of semi-elliptical cracks. This finite element program uses solid, isoparametric, twenty noded, curved hexahedral elements in conjunction with the superelement program NV336.

Further details of the use and content of this suite of computer programs may be obtained from the user manuals as published by DNV. Some examples of the use of these analysis programs, as reported in the literature will be described in Chapter 4.

3.3.2 Lloyd's Register of Shipping

LLoyd's Register of Shipping has provided services for certification, classification, independent design appraisal and quality assurance of offshores structures since 1971. Extensive stress analysis, dynamic and fatigue analysis are carried out by this organization on all types of offshore structures including fixed steel platforms, jack-ups, semi-submersibles, crane and pipelaying barges, articulated columns for flaring and loading, concrete platforms, tension leg platforms and large floating storage and production vessels in accordance with accepted standards and guidelines. To carry out these analyses a number of purpose built computer analysis packages have been developed. These packages were built around a structural analysis program, MSC NASTRAN which is the core of the system.

MSC NASTRAN (92) is a general purpose finite element program designed to analyse the behaviour of elastic structures under a range of loading conditions using the displacement method. The capabilities of this computer package include:

(i) Static analysis (92) using concentrated and distributed loads; thermal expansion and enforced deformation.

(ii) Dynamic analysis (93) of transient loads, steady state harmonic loads and random excitation.

(iii) Determination of real and complex eigenvalues.

The element library available in NASTRAN includes more than fifty finite elements. The elements include 1-, 2- and 3-D elements, scalar elements, axisymmetric elements, rigid elements, mass elements, fluid elements and heat transfer elements. A description of these elements is given in (92). Nastran has an excellent capability for the solution of very large problems in

an accurate and efficient manner. In some cases static and dynamic problems with more than 100,000 degrees of freedom have been solved using this system. Nastran also has a superelement capability for the solution of problems that are too large to be conveniently analyzed in a single execution.

The graphics capability of the Lloyd's system is provided by a program LR ICON. This program is a fully interactive system designed by Lloyd's to create and check data for finite element analyses by use of a display terminal, light pen and keyboard. This graphics input system is common to all the Lloyd's analysis packages and has an interface program to the NASTRAN package. The output from the analysis packages uses high speed drum plotters or Tektronix hard copy units. Using these devices hard copies of structural plots, stress contours, critical wave loading positions and response spectral plots may be obtained. Examples of structural model plots are shown in Figs. 3.1 and 3.2.

The analysis packages in the Lloyd's system are shown in Table 3.2. The individual programs in each package are listed in (94,95). A brief description of the capabilities of the more significant analysis packages is given below.

LR LOPS: The analysis of fixed piled steel structures (96)

This is a computer analysis package for the structural analysis and fatigue life estimation of fixed steel platforms. The data for the structural model is created using the interactive graphics system ICON. There are various options within the system for modelling the waves including Airy linear, Stokes's 5th order and Munk wave theories and Dean's stream function. The forces produced by the waves are calculated using Morison's formula for small diameters. The programs also calculate buoyancy loads and the current effects which vary with depth. Various wave directions may be considered and the position of the wave to maximize horizontal load or overturning moment may be determined at any particular level. The analysis and calculation of internal forces are carried out using NASTRAN. Other programs in this package also carry out stress checks according to AISC criteria, punching shear checks according to API criteria, hydrostatic checks and the calculation of stress concentration factors for tubular joints. Fatigue analysis may also be carried out using programs in this package. A variety of different waves representative of the environment are used to give a spectrum of loading and the

expected fatigue life is estimated by calculating stress variations at positions around the joint.

The programs use the Palmgren-Miner law of cumulative damage for estimation of life. Other analyses possible using this package are earthquake analysis and pile analysis in which the non-linear soil properties are modelled using a piecewise linear analysis procedure of NASTRAN. The individual programs in LR LOPS are listed in (97) and a flow chart of the analyses possible with LR LOPS is shown in Fig. 3.3.

LR LOCS: Analysis of Concrete Gravity Structures

LR LOCS is a package of programs for the analysis of concrete gravity structures. This package uses NASTRAN for structural analysis and is similar to LR LOPS in construction. In these analyses the diameter of the members is quite large and therefore the wave loads are evaluated in a program that utilizes wave diffraction theory. The programs in this package also facilitate, elastic analysis of reinforced concrete sections, limit state analysis, shell intersection analysis and post tensioning calculations of elongations in reinforcements.

The structures analyzed by this package are large and cyclic symmetry is used extensively to reduce the size of the finite element model. An example of a segment of a concrete structure model is shown in Fig. 3.2.

LR LOTS: Analysis of Tension Leg Platforms

This system is designed to analyse the motions and loads on a tension leg platform in regular and irregular seas. The maximum probable stresses are calculated using NASTRAN. The loadings on the structure during motion due to hydrodynamic and hydrostatic forces, gravity, inertia, buoyancy, wind, current and tension leg forces, can be taken into account. All six degrees of freedom are included in the calculation of the motion.

LR LOADS: Analysis of Semi-submersible Structures

The LR LOADS system of programs permits a modified dynamic analysis or a spectral analysis of the motions of semi-submersible structures. In the modified dynamic anlysis approach the structure is represented by an FE model and an appropriate wave theory chosen. The critical waves on the structure are determined and the corresponding loads on the structural elements calculated. The equipment loads, are also calculated and the loaded structure is

analysed using NASTRAN to obtain the stresses in the structural elements. These stresses are then checked using the AISC permissible stress levels to ensure adequate safety factors against yielding or buckling. The spectral analysis approach is designed to analyse the motions of a semi-submersible in regular or irregular seas. As before the motions are calculated using all six degrees of freedom and the loading due to hydrodynamic and hydrostatic forces, inertia, buoyancy, mooring and towing forces, wind, and current are taken into account in the calculation. Again the NASTRAN finite element package is used to calculate the maximum probable stresses.

LR LADS: Load Response Spectral Analysis of Steel Structures

LR LADS is an analysis package designed for spectral dynamic fatigue analysis of steel structures that are installed in deep water. The fatigue life estimation of the steel joints in these structures is based on the probability distribution of stress ranges evaluated from a dynamic spectral analysis. The natural frequencies of the structure and the characteristic frequencies of the wave load spectra are used to calculate the response transfer functions per unit wave height. The fatigue analysis program in this package uses these transfer functions, the stress concentration factors, the seastate scatter diagram and a directional probability in the calculation of stress spectra and hence the estimation of fatigue life. The cumulative damage is calculated using the Palmgren-Miner cumulative damage rule and the appropriate S-N curves.

SEFACS: Stochastic Extreme Value and Fatigue Analysis of Concrete Structures

This package is an equivalent system to the LR LADS package developed for the spectral dynamic analysis of concrete gravity structures subjected to random wave excitations. The structure and foundation system is represented by finite element modelling and the global response of the structure to wind and wave action is obtained using the capabilities of NASTRAN. The spectral analysis follows the general procedures outlined above, and in section 2.5.

Analysis of Jack-up Structures

The analysis of jack-up structures is carried out using sections of the packages described above. The lattice leg jack-up structures may be idealized using the graphical capabilities of the Lloyd's system. The wave loads are calculated as described in the LP LOPS package and the critical leg load-

ing cases may be determined using the static and dynamic capabilities of the NASTRAN system. If necessary the dynamic effects may be investigated using appropriate portions of the spectral analysis package LR LADS. As before the stresses may be checked against the AISC and API recommended values.

LR FEEPHO: Analysis of Soil

The analysis of soils is carried out by the program LR FEEPHO and associated programs. The non-linear behaviour of soils can be analysed in the two dimensional plane strain and axisymmetric versions of the static, linear elastic, perfectly plastic finite element program LP FEEPHO. The output from this system are contours of stress and displacement. Pile-soil interaction and pile drivability may be studied using programs in this package. Also groups of piles may be modelled and their behaviour in uniform and stratified soils investigated.

Further information of the packages for the analysis of articulated columns, the analysis of risers and the analysis of mooring systems may be obtained in (95).

3.3.3 American Bureau of Shipping ABS

The Bureau of Shipping is the American classification and certification authority for offshore structures. The Bureau classifies a structure as to its structural and mechanical fitness for a particular use or service in accordance with its rules and standards. To assist in the classification process the ABS has developed or used a series of computer programs Table 3.3. The analysis procedures for a fixed offshore structure are represented by a flowchart as shown in Fig. 3.4. The geometric and structural FE model of the drilling platform or floating unit is generated using the program RIGGEN. This program can generate a two or three dimensional model consisting of beams, rods, bending and membrane plate finite elements. The structural model can then be displayed using the program RIGPLOT so that the input data can be thoroughly checked. The current and wave forces on the structure are calculated using the program RIGLOAD. This computer program has the capability of representing the waves using either the Airy wave theory, Stoke's 5th order wave theory or the Cnoidal or Solitary wave. The forces on the members are calculated using Morison's equation. This process can be repeated for different waves and different angles of attack. The maximum total force or moment on the structure can then be determined. The analysis of piles, supporting

structures subjected to lateral and axial loads can be carried out by the program BMCOL, in the ABS system. The program performs a finite difference solution of a straight, two dimensional linearly elastic and symmetrical beam resting on non-linear supports. The non-linear support properties are described by the input of the P-Y (load-deflection curve) that characterises the supporting soil.

The analysis capability in the ABS system is supplied by one of the general purpose finite element programs, NASTRAN, DAISY or SAP IV, (99). NASTRAN is described in section 3.3.2. DAISY (Displacement Automated Integrated System) is a general purpose structural finite element program which performs linear elastic structural analyses of complex 2- or 3-D structures. The element library in DAISY includes rods, beams, triangular and quadrilateral membrane and bending elements. The analysis is confined to statically applied generalized forces and or thermal loads.

SAP IV (Structural Analysis Program) performs linear elastic finite element analysis of 3-D structural models. A special feature of the system is its dynamic analysis capability. This includes mode shapes and frequencies, dynamic response analysis for arbitrary time dependent loads and response spectrum analysis. Further details of the capabilities of the DAISY and SAP IV finite element packages may be found in (99,100).

The ABS system of programs include the plotting programs GENPLOT (General Plot), SAPLOT (SAP Plot) and DAISYOUT which plot the finite element model, perform checks for data consistency and plot the output displacement of the loaded models.

Other programs in the ABS library of particular interest in offshore structure analysis are ABS/SEMISUBMERSIBLE and CAISSON. ABS/SEMISUBMERSIBLE is a collection of three sub-programs which compute the wave-induced loads and motions of a twin-hull semi-submersible platform. These programs yield the amplitudes and phase angles of the motions of the six degrees of freedom, the pressure distribution, the shear force, bending moments and mooring forces on the structure. CAISSON is a program which calculates the wave and current loads on a large gravity structure. This structure consists of a large volume caisson and a superstructure composed of small diameter tubular members. The wave and current loads on the base caisson are calculated using diffraction theory while the wave and current loads acting on the superstructure, which include both drag and inertial terms are calculated either by linear or 5th order wave theory of Morison's Equation.

A listing and description of the many programs in the ABS computer programs library is given in (98).

3.3.4 SACS (Stuctural Analysis Computer System)

The SACS system of computer programs was developed by Engineering Dynamic Inc. of New Orleans for the analysis of offshore structures (101). The system may be used for the analysis and design of fixed steel platforms, semi-submersible rigs, drillships and barges. The fully integrated program modules are listed in Table 3.4 and a general flow diagram of the SACS system is shown in Fig. 3.5. The system may be divided into three sections, input generators (pre-processors), solution programs and post-processors. The input generators create a common input data file for the solution programs which in turn create a common output file which can be accessed for the post-processing of results. A short description of the program modules in the SACS system follows.

AUTOGEN

This program module provides automatic generation of input data for the SACS solution programs. Structural regularity and symmetry are utilized to minimize input data. It is possible to create finite element models of a complete structure or more detailed models of a complex joint.

SEASTATE

SEASTATE generates static and dynamic load data simulating the offshore environment at the site of the structure. Loads due to waves, buoyancy, wind, current and dead loads may be calculated. The program generates wave data using Airy, Stokes's 5th order theory or stream function wave theory or the user may specify any arbitrary wave in either velocity or pressure field input. The wave is automatically stepped through the structure and the position giving maximum base shear or moment is selected for futher analysis. Buoyancy of non-flooded members are calculated automatically and the flooding or non-flooding of members may vary with each load case. This module also computes the wave forces for dynamic response analysis.

LAUNCH/FLOTATION/TOW

These programs provide analyses of jacket motions and loadings during towing, launching and installation on site. The launch analysis provides time

histories of the barge and jacket of specified times during launch. The time history analysis is performed with variable step integration. The flotation analysis allows the user to determine jacket position and stability during the flotation and up-ending procedure. The TOW program describes the dynamics of the model during a towing sequence and produces a data input file for further analysis by the SACS system.

SACS III

SACS III is the core of the static analysis portion of the SACS system. It is a large capacity general purpose finite element program for linear elastic static structural analysis. The element library includes beam, shell and plate elements. The program also has the additional capabilities of an automatic redesign feature, a hydrostatic collapse analysis, and a weight and materials take-off feature. To further enhance the range of capabilities of SACS III several additional programs can be automatically coupled to it to carry out more specific and complex analyses. These programs are described below.

PSI

The pile-structure interaction program performs the analysis of a linear pile supported structure, couple to non-linear piles and subjected to static loads. The solution is interactive and the PSI program may be coupled automatically to AUTOGEN, SEASTATE, STRESS CHECK, JOINT CAN and STRUMP. The output of PSI includes the linear structural analysis and the stress analysis of the piles.

DYNPAC

DYNPAC generates the mode shapes, periods and modal stress for a structural model generated by AUTOGEN. Automatic lumped or consistent mass generation including fluid added mass is available. The user may designate flooded or non-flooded members and the output from the program is compatible with the display post-processor STRUMP.

SPECTRE

The SPECTRE program performs a spectral and time history response analysis of a structure. Earthquake analysis forced response analysis due to waves and current may be performed. Earthquake analysis may be carried out

either spectrally or in the time domain. For the spectral analysis the API earthquake response spectrum may be accessed automatically or the user may define a response spectrum. For time history analysis non-linear fluid effects may be included and the program provides a variable step integration procedure which minimizes computer time while maintaining accuracy. The forced response analysis due to waves and currents is performed with input from the SEASTATE program. Non-linear fluid forces, including the effects of structural motion are included in the analysis. The output from this program is compatible with the SACS post-processors.

COMBINE

The COMBINE program has the capability of combining the results of a large number of basic load cases in a linear fashion to form new load case results files. This program may be used to combine the modal responses in forced vibration analysis and for symmetry/antisymmetry analysis of large structures. This facility is a very useful tool in finite element structural analysis.

MTO

The materials take-off program generates lists of materials by structural type, calculates surface areas and weld volumes and performs cost analysis of the materials required. This program can be coupled with the redesign capability of the SACS III analysis program to provide rapid cost data for various structural configurations during the design process.

STRESS CHECK

This program is a SACS system post-processor used to perform unity checks on the stresses in all plate and beam elements according to the AISC and API codes. Detailed summary reports of the stresses are available as output.

JOINT-CAN

This post-processor performs punching shear analysis according the API code. Joints having multiple intersecting members are analysed automatically and joint cans may be re-designed by modifying chord wall thickness by specified increments.

FATIGUE

FATIGUE is a SACS post-processor program used to operate on the common solution file provided by either a linear static SACS III analysis or a dynamic DYNPAC/SPECTRE analysis. The program automatically calculates the stress concentration factor for joints and estimates the fatigue damage at eight points around the joint using built in S-N data. The program has the options of including dynamic amplification of stresses, allows the user to input different S-N data or to override the SCF at specified joints. This program also has the capability of redesigning tubular joint thicknesses to satisfy fatigue requirements. The fatigue damage is estimated using the Palmgren-Miner law.

STRUMP

STRUMP is the plotting and display module of the SACS system and it may be used as a pre-processor to display the geometric model to aid in data-input checking or as a post-processor to display the static deflections or dynamic mode shapes of the loaded structure. The program has an extensive plotting capability that allows variation of plot size, selection of planes or volumes of the structure to be plotted, rotation of views, hidden line plots and plotting of the superimposed deflected and undeflected structure.

3.3.5 ASAS Offshore Suite of Programs

The ASAS offshore suite of programs was developed by Atkins Research and Development and has been used extensively by engineers for the analysis of offshore structures particularly those in the North Sea. The programs in the offshore analysis package are listed in Table 3.5 and a flowchart showing the sequence of analyses possible is shown in Fig. 3.6.

The heart of the system is the finite element program ASAS. The ASAS program together with its many pre-and post-processors, listed in Table 3.6, is a comprehensive general purpose finite element system (102)-(106). The main program incorporates facilities for linear elastic static stress analysis, natural frequency analysis and heat conduction analysis. The system has an extensive library of element types and a wide range of loading and boundary condition types. A comprehensive series of data checking procedures which check input data for errors and consistency is also included. The pre- and post-processors cover a wide range of capability from mesh generation and graphic display to load case combination and stress contour plotting.

Using the ASAS structural analysis system and the programs in the ASAS offshore package it is possible to carry out a comprehensive series of analyses on offshore structures, which will be outlined below.

(i) Design Wave Analysis

In this analysis the designers initial member sizing and layout is tested and modified to meet the service requirement. In this case the main requirement of the analysis is to ensure that the stresses in the structure remain within code allowable limits when it is subjected to a maximum expected storm wave. The basic structural layout is generated using ASAS and its pre-processors e.g. MESHMOD. This data is checked visually using ASDIS, the display module. The wave current and wind forces for each loadcase are generated using ASASWAVE. This program selects an appropriate wave theory, from Airy, Cnoidal 1st, Cnoidal 2nd, Stokes's 5th wave theories or stream function theory, calculates the water particle velocities and hence computes the forces on the members using Morison's equation. Buoyancy, internal water mass and increase in member diameter due to marine growth are all considered. ASASWAVE calculates the wave, wind and current loading on steel jacket structures for subsequent static, dynamic and fatigue analyses.

The analysis of the structure is performed by the static analysis capability of ASAS, in which the wave loads case and dead load case, (deck loads etc.), are applied separately to the structure to obtain the deflected shape, axial loads, bending moments and stresses in the members. Combination of the dead load case with each wave load case is carried out by ASASLOCO. Stress checks to AISC, API, Department of Energy (UK) or DNV codes is performed on all tubular members by program JACKAS. This program also checks each member for local buckling, overall buckling, highest direct stress and punching shear and in the event of an overstress suggests a specific change in member size.

(ii) Dynamic Analysis for Natural Frequencies and Mode Shapes

The basic structural description and environmental loading as required for the design wave analysis is required as input for this analysis, together with information on the mass of all elements, added mass due to marine growth and flooding of members. These masses are calculated by ASASMASS. The natural frequencies and mode shapes are computed using the dynamics solution capacity of ASAS. The subspace iteration technique can be used to deter-

mine the eigenvectors for all degrees of freedom in the structure that give an accurate solution and reliable stress recovery. For large problems, condensation methods may be used to reduce the size of the problem but the resulting solution is adequate only if deflections or overall dynamic effects are required. In an offshore jacket structure the first 20 or 30 mode shapes are sufficient. The mode shapes may be displayed graphically using the plotting routine ASDIS. An example of mode shapes output by ASDIS is shown in Fig. 2.5.

(iii) Dynamic Design Wave Analysis

As stated in an earlier section of this report in the true dynamic situation the peak deflections and stresses may be 10% greater than those obtained in a static analysis, therefore, the dynamic response of the structure to the passage of the design wave should be obtained particularly for structures in deep water. This analysis may be performed using a transient response analysis following a natural frequency analysis. The transient response analysis is carried out by ASASRESPONSE and the response is computed by direct integration using the inherently stable and efficient Newmark β alogorithm. The wave input is produced by ASASWAVE with the wave forces evaluated for a number of increments of phase throughout the wave cycle. The output stress file is of the same format as the output from a static analysis and therefore it may be post-processed by the stress checking program JACKAS.

(iv) Fatigue Analysis

The ASAS offshore package permits two approaches to fatigue analysis, i.e. a deterministic or a spectral approach.

(a) Deterministic Fatigue Analysis

In this analysis it is assumed that the effect of the various sea-states the structure must endure can be represented by a large number of regular wave trains, each of suitably long duration. For each wave train the steady state, structural dynamic response is calculated for all the modes previously obtained by natural frequency analysis. These responses are computed in nodal displacement form and summed together to give the complex steady state displacement of the structure. The stress amplitude and the duration of all representative wave conditions is obtained and hence a stress history for each point on the structure is calculated. A stress concentration factor calculated by empirical formulae may be applied to each calculated

stress to convert them to hot-spot stresses at the welds. Finally the S-N data curves for the material are used to find the damage increments for each wave train and the accumulated damage is obtained using the Palmgren-Miner rule.

(b) Spectral Fatigue Analysis

A spectral fatigue analysis may be performed using the FATJACK program. The seastates are represented by relationships such as the Jonswap, Scott-Wiegel or Pierson-Moskowitz spectra and the stress-transfer functions are obtained from the results of dynamic response to a series of regular wave trains. The stress response spectrum is obtained using this information. The fatigue damage is obtained using the material fatigue data and the relative occurance of the sea state as outlined in section 2.5. The process is repeated for other wave direction sectors and the total damage calculated using the Palmgren-Miner rule as before. The printed output from the program gives the fatigue life expressed in years for each of eight inspection points at each end of the tubular member and those values below a given design target are flagged.

(v) Launch and Up-ending Analysis

The analysis of jacket launch and up-ending is performed by the programs ASASLAUNCH and ASASFLOAT (105). ASASFLOAT computes the position dependent hydrostatic loading on a free floating structure. The program requires a hydrostatic model of the structure then the equilibrium and stability of the structure can be calculated. ASASFLOAT allows for the trimming of the structure by controlled flooding of members during upending, and checks the stability at each stage. The program ASASLAUNCH computes the time dependent trajectory of the jacket and barge during the launching sequence. This program requires hydrostatic and dynamic models of each structure together with a description of the track system used for launching. This program also calculates the hydrodynamic and hydrostatic forces on each member at every time step during the launching sequence. These member forces are summed to determine the total applied loading on each structure. The loads at any time step may be transferred as a loadcase to the ASAS program and a stress analysis of the structure can be carried out. This program is also used in the up-ending sequence to perform a time history analysis with the rates of flooding of various members and buoyancy chambers as input.

The output from ASASLAUNCH and ASASFLOAT is presented in graphical form using the program C-gul. This program has the capabilities of visualization of structure trajectories, zoom, multi-frame overlays and pictorial perspectives. Examples of the output of C-gul are shown in Figs. 3.7 to 3.11.

(vi) Pile Analysis

The simulation of the behaviour of single and groups of piles in soils with non-linear characteristics is performed using the programs SIMPILE and GROPILE. The pile is idealized as a series of beam elements and the soil is simulated as a series of springs with associated plastic components. Elastic-plastic soil properties are input for both transverse and longitudinal directions and the program uses an iteration technique to find the equilibrium position of the pile. The analysis of a group of piles is more involved because the interaction between piles must be considered. The output from these programs gives the deflected shape, internal stresses and AISC stress ratios throughout the length of each pile.

(vii) Detailed Joint Stress Analysis

A key area in offshore structures requiring detailed stress analysis is the tubular joint region where the stress distribution and load paths are very complex. ASAS has a wide range of capability for analysis of these regions. The general curved shell family of isoparametric finite elements are the elements recommended for this application. These elements exist in both triangular and quadrilateral form and combine good performance with relatively few degrees of freedom. They also have the ability to be severely distorted in areas of non-essential interest to promote economy.

The input for these analyses may be produced by the program TUBEGEN (106). This program is an interactive mesh generation program for tubular joints that produces finite element meshes for T, Y, X and K type joints. An example of an FE mesh produced by TUBEGEN is shown in Fig. 3.12. The output from a joint stress analysis can be in numerical form or can be presented graphically as stress contours by the ASAS post-processor ASPECT, Fig. 3.13.

3.3.6 MARCS: Marine Structures Computer System

The MARCS Marine Structure Computer System was developed by the Offshore Construction Industry Department of Synercom Technology Inc. of Texas.

This system consists of the programs listed in Table 3.7 and is described briefly below.

AUTOPLAT

AUTOPLAT, is a program that automates the production of structural data for the subsequent analysis of an offshore structure. This program generates the structural model, prepares graphical plots of the structure and generates a "bill of materials". The user may select any section of the structure for plotting and the bill of materials includes the total length and weight of each member and the total weight of the structure. This program also includes an extensive error checking facility to assure the accuracy and completeness of the input data.

SEALOAD

The SEALOAD program calculates the wave, current, wind and gravity loads on an offshore structure and prepares the load data set for analysis by STRAN. The user may specify either the Airy, Sine, Solitary or Stokes's 5th order wave theory and may also choose the wave angle of attack. The program selects the wave position for maximum shear or overturning moment for a given angle of attack. The effects of dead loads, and flooded or filled members on buoyancy are automatically considered. The program also calculates the loads due to wind on deck structures and members out of the waves. The output from this program is a data file for direct input into the structural analysis program STRAN.

STRAN

STRAN, is the structural analysis program of this offshore structures analysis package. It utilizes the substructure approach and therefore can handle very large problems with extreme efficiency. A linear analysis of the structure with non-linear boundary conditions may be performed and a stress evaluation in accordance with the AISC or API codes is carried out. A punching shear (API) calculation may also be obtained. The dynamic capability of the STRAN program is a recent enhancement. Using this facility, natural frequency and modes shapes of the structure may be computed using the subspace iteration method. A steady state response analysis of a structure subjected to ocean wave loading may be performed by STRAN used in conjunction with SEALOAD. The output of STRAN includes stress summaries, joint deflections and

rotations, reaction forces and moments, member forces and moments and punching shears. An example of a mode shape calculated by STRAN is shown in Fig. 3.14.

LAUNCH-3D

This program is a 3-D motion simulation of both jacket and barge during launch and up-ending. Using this program the designer is able to investigate the behaviour of the jacket as it slides from the barge, the jacket stability as it floats in the water and the flooding sequence of jacket members necessary to up-end the jacket.

FATIGUE

This program, when used in conjunction with the wave load program SEALOAD and the analysis program STRAN provides a means for determining the fatigue life of a structure. The fatigue damage may be computed using the probabilistic or the deterministic approach. Stress concentration factors for the various types of joint are calculated using empirical formulae and the S-N data curves may be pre-defined or changed by user input. The wave statistics may be defined by Pierson-Moskowitz or Jonswap spectra, or the user may input the spectrum for each seastate. The analysis is carried out as previously described and the output gives the critical fatigue positions around the welds, the maximum hotspot stress for a deterministic analysis, the cumulative damage ratio and the fatigue life.

PILEAN

The PILEAN program is a pile-soil analysis program that computes the lateral deflections and bending moments of piles embedded in a soil having non-linear material properties. The piles may have uniform or variable cross-sections as a function of depth and as many as 50 piles can be considered in a single application of the program. The output presents the pile deflections and bending moments as a function of depth.

SEASAP

This program has been replaced recently by the enhanced dynamic capabilities of the analysis program STRAN. SEASAP is a pre-processor for the SAP IV structural analysis program enabling MARCS users to use this program for dynamic analysis of offshore structural models. In essence SEASAP rearranged the MARCS input data files into the SAP IV input format. Frequencies, modes

shapes and response history analysis can now be computed using the SAP IV program.

3.3.7 Other Programs

Many other general purpose finite element programs exist that could be used for the structural analysis of offshore structures. These programs include ANSYS, EASE2, MARC, STRUDL, STARDYNE and SUPERB. These programs are general purpose, state-of-the-art, commercial finite element programs. However, they do not have specific pre- and post-processing capabilities for application to the offshore industry. One exception that does have a wave loading capability is the STARDYNE program which has an associated program WAVE 4 that computes hydrodynamic forces on tubular and or circular members.

These general purpose FE programs have a wide range of capabilities for structural analysis, a selection of them are shown in Table 3.8. Further details of these computer packages may be obtained from review references such as (107,86).

3.4 PROGRAM AVAILABILITY

Most of the programs described in this report can be purchased or leased and installed in-house on a suitable main-frame computer, however, a more cost effective method of access for the occasional user is through a computer bureau. The availability of the offshore computer packages is given in Table 3.9. The addresses of these bureaux and the owners or developers of the package are given in Appendix I.

TABLE 3.1 Finite element types: SESAM 69

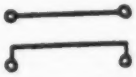
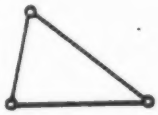
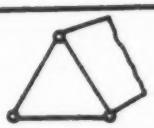
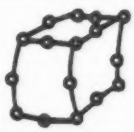
Element	d. o. f. per element	Description
	2	1. Straight bar element. Constant strain. 2. Potential element Linear
	6 3	1. Triangular membrane element. Constant strain, orthotropic. 2. Potential element linear.
	8	1. Quadrilateral membrane element. Linear normal stresses, constant shear stress, orthotropic.
	12	1. Centric or eccentric beam element with shear deflection
	12	1. Beam elements including shear deflection. Several cross-section types. Variable rigid lengths and rotational fixity of the nodes.
	12 12	1. Straight tube element 2. Curved tube element
	15	1. Triangular thin shell element. Constant strain membrane action, linear moment distribution. Orthotropic.
	20	1. Quadrilateral thin shell element. Membrane action for quadrilateral membrane element. Bending action by two layers of triangular bending element. Orthotropic.
	6	1. Triangular ring element. Constant strain. 2. Potential element. Linear.
	60 20	1. Isoparametric solid element. Parabolic shape function. 2. Potential element. Parabolic shape function.
	40	1. Superparametric thick or thin shell element. Parabolic shape function.

Table 3.2 Offshore structure analysis packages. Lloyd's register of shipping.

Analysis Package	Description
LR LOPS	Analysis of Fixed Piled Steel Structures
LR LOCS	Analysis of Concrete Gravity Structures
LR LOTS	Analysis of Tension Leg Platforms
LR LOADS	Analysis of Semi-submersible Structures
LR LASS	Analysis of Articulated Columns
LR LADS	Load Response Spectral Analysis of Steel Structures
SEFACS	Stochastic, extreme value and fatigue analysis of concrete structures
LR LARS	Analysis of Risers
LR FEEPHO	Analysis of Soils

Table 3.3 ABS: Computer programs for the Analysis of Offshore Structures.

Program	Description
RIGGEN	Model Generation
RIGPLOT	Model Plotting and Checking
RIGLOAD	Wave and Current Loading
BMCOL	Analysis of Piles
NASTRAN	Structural Analysis
DAISY	Structural Analysis
SAP IV	Static and Dynamic Analysis
ABS/SUBMERSIBLE	Motion Analysis of Semi-submersibles
CAISSON	Wave and Current Loads on large gravity structures
GENPLOT	Plotting programs for graphical output
DAISYOUT	
SAPLOT	

Table 3.4 SACS: Computer programs for the Analysis of Offshore Structures.

Program	Description
AUTOGEN	Automatic generation of input data
SEASTATE	Generates wave, current wind and dead load input data
LAUNCH/ FLOTATION	Analysis of jacket launch and up-ending
SACS III	Linear elastic, static structural analysis program
PSI	Pile Structure Interaction
PILE	Pile Analysis
DYNPAC	Generates dynamic characteristics for a structural system
SPECTRE	Spectral and Time History Response Analysis
STRESS CHECK	Stress checks against AISC and API codes
JOINT CAN	Punching Shear Analysis
FATIGUE	Fatigue Damage Calculation
STRUMP	Plotting Program
MTO	Materials Take Off
TOW	Towing Analysis

Table 3.5 ASAS - Offshore Suite of Programs

Program	Description
ASAS	Static and Dynamic General Purpose Finite Element Program
ASASWAVE	Computation of Wave Current and Wind Forces
ASASMASS	Computation of mass due to marine growth, non structural mass, flooded mass etc.
ASASRESPONSE	Transient Response Analysis
JACKAS	Stress Checking
FATJACK	Spectral Fatigue Analysis
ASASFLOAT	Hydrostatic Stability of free floating strucute
ASAS LAUNCH	Analysis of Jacket Launch
C-gul	Plotting program for ASASFLOAT and ASASLAUNCH
SIMPILE	Single Pile Analysis
GROPILE	Group Pile Analysis
TUBEGEN	Mesh Generation of Intersecting Cylinders

Table 3.6 ASAS pre- and post-processors

(i) Pre-Processors

(a) ASBAND - Data optimization program to reduce the equation bandwidth and hence improve efficiency of the solution.

(b) INSTRUCT - A free format data input program.

(c) MESH3/MESH4 - Automatic mesh generation programs to create triangular or quadrilateral two dimensional elements.

(d) MESHMOD - Interactive mesh modification program.

(ii) Graphic Display Module

ASDIS - Graphic display program used to display the original structure, and after analysis, to plot the deformed shape of the model.

(iii) Post-Processors

ASASLOCO - A general purpose post processor to select, factor and combine the results from previous ASAS analyses.

ASPECT - An interactive graphics program for stress contour plotting. Used as a post-processor of the results from an ASAS analysis to visually identify the stress distribution within a structure.

AXIST - A post-processor program to calculate average nodal stresses, principal stresses, Von Mises stresses and Tresca stresses for axisymmetric elements.

BRICKST - A post-processor program to calculate average nodal stresses, principal stresses, Von Mises stresses and Tresca stresses for the family of general three-dimensional isoparametric elements.

PLATEST - A post-processor program to calculate average nodal stresses, prinicpal stresses and Tresca stresses for the family of flat plate elements.

SHELLST - A post-processor program to calculate average nodal stresses and principal stresses for the top, bottom and mid-plane surfaces of the family of shell elements. The equivalent Tresca yield stress and Wood Armer stresses and moments may also be calculated.

FRAKAS - Post-processor that calculates the energy release rate values, for crack problems, using the virtual crack extension method.

Table 3.7 MARCS - Marine Structures Computer System

Program	Description
AUTOPLAT	Structural Data Generator
SEALOAD	Wave Loading Program
STRAN	Static and Dynamic Structural Analysis Program
LAUNCH-3D	Three dimensional platform launch, flotation and up-ending analysis program
FATIGUE	Fatigue Damage Analysis Program
PILEAN	Pile/Soil Analysis Program
SEASAP	Sea State Structure Pre-processor for SAP

Table 3.8. Types of analysis.

ANALYTICAL CAPABILITY		PROGRAM					
		EASE2	STARDYNE	NASTRAN	ANSYS	MARC	SUPERB
LINEAR STATICS	MECHANICAL LOADS	*	*	*	*	*	*
	TEMPERATURE LOADS	*	*	*	*	*	*
	EULER BUCKLING			*		*	
	INERTIA RELIEF			*			
DYNAMICS	MODE/FREQUENCY	*	*	*	*	*	
	FREQUENCY RESPONSE		*	*	*		
	TRANSIENT RESPONSE	*	*	*	*	*	
	SHOCK SPECTRA	*	*		*		
	RANDOM RESPONSE		*	*			
	NONLINEAR TRANSIENT			*	*	*	
NONLINEAR STATICS	NONLINEAR BUCKLING					*	
	LARGE DISPLACEMENT					*	
	PLASTICITY			*	*	*	
	CREEP				*	*	
	VISCOELASTICITY			*		*	
	LARGE STRAINS					*	
HEAT TRANSFER	STEADY STATE			*	*	*	*
	TRANSIENT			*	*	*	
SUBSTRUCTURES (SUPER ELEMENTS)	STATIC		*	*	*		
	DYNAMIC		*	*	*		
	CYCLIC SYMMETRY			*			
MISCELLANEOUS	FRACTURE MECHANICS				*	*	
	FLUIDS			*	*	*	
	ELECTRIC CIRCUITS				*		
	OPTIMIZATION			*			
	ACOUSTIC CAVITIES			*			
	FATIGUE DAMAGE				*		

Table 3.9 Program Availability

Program	Availability
DNV/SESAM-69	A.S. Computas (NORWAY)
Lloyd's	Lloyd's Register of Shipping
ABS	American Bureau of Shipping, New York
SACS	C.D.C
	S.I.A. Ltd.
	United Computing Systems
	Bocing Computer Services
	University Computing Company
	Information Systems Design
	Multiple Access (possible)
ASAS-OFFSHORE	CDC (ASAS only)
	United Computing Systems
MARCS	CDC
	United Computing Systems
NASTRAN	CDC
	Multiple Access
	Comshare
ANSYS	CDC
	Multiple Access
	Comshare
MARC	CDC
EASE2	CDC
SUPERB	CDC
	Multiple Access
	Comshare

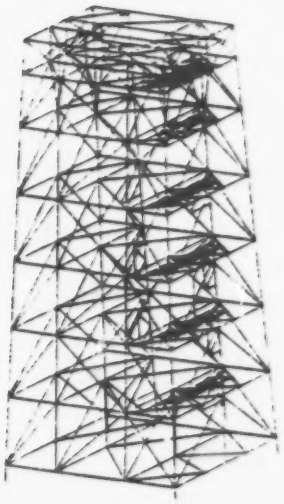


Fig. 3.1. FE model of fixed steel platform using LOPS program (neg 6712)

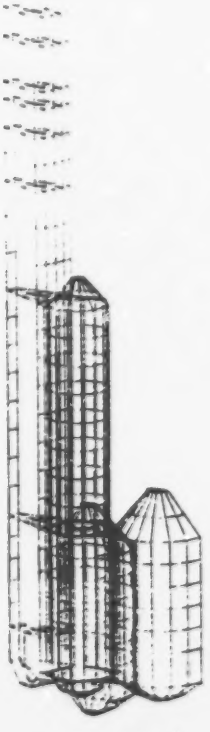


Fig. 3.2 FE model of a section of a concrete structure. (neg 6692)

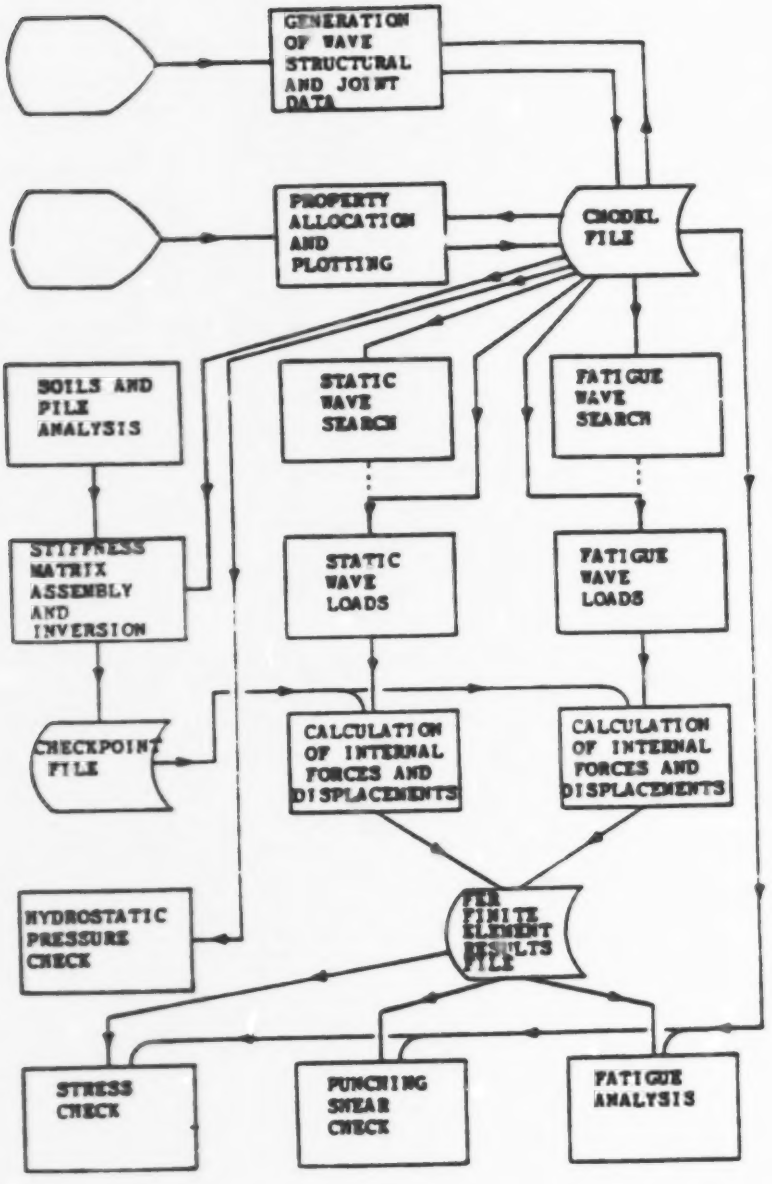


Fig. 3.3. Simplified diagram of the LR LOPS system. (neg 6693)

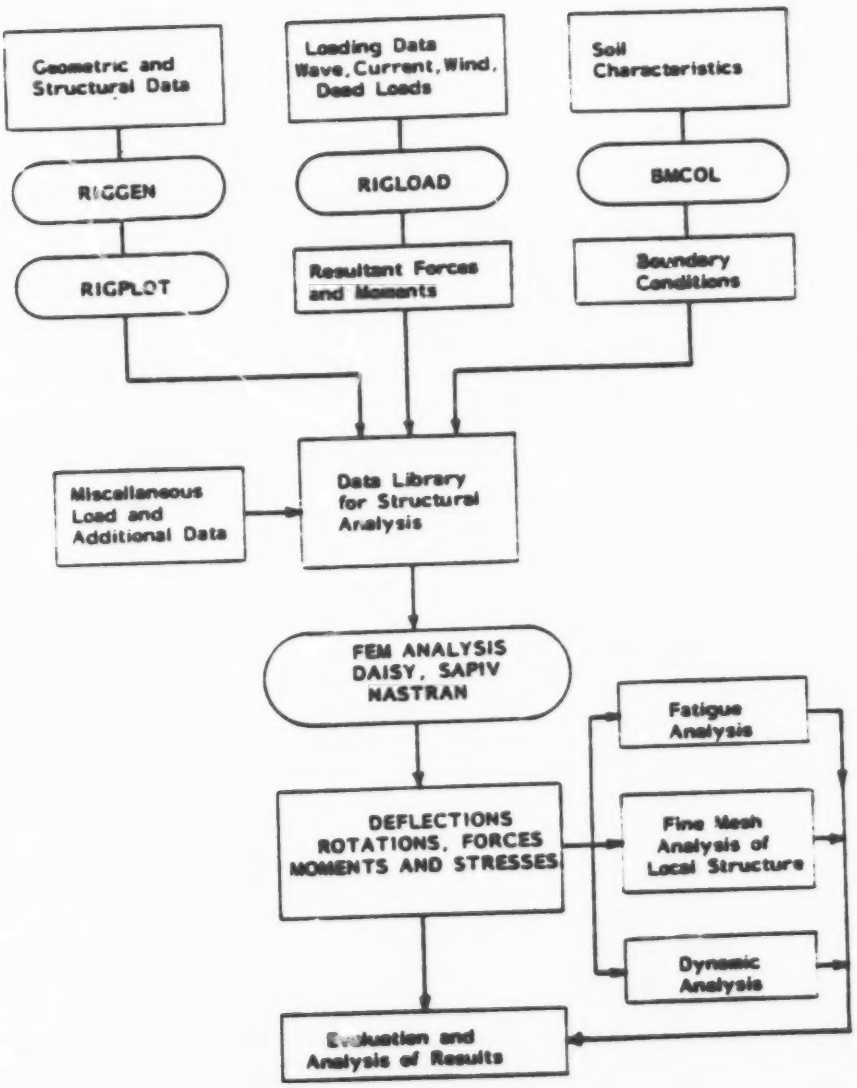


Fig. 3.4. ABS: Procedure for analysing fixed platforms. (neg 6699)

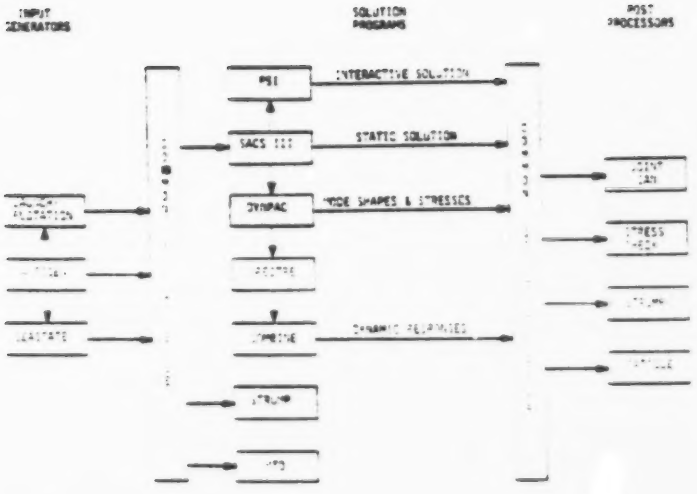


Fig. 3.5. Structural analysis computer system. (neg 6713)

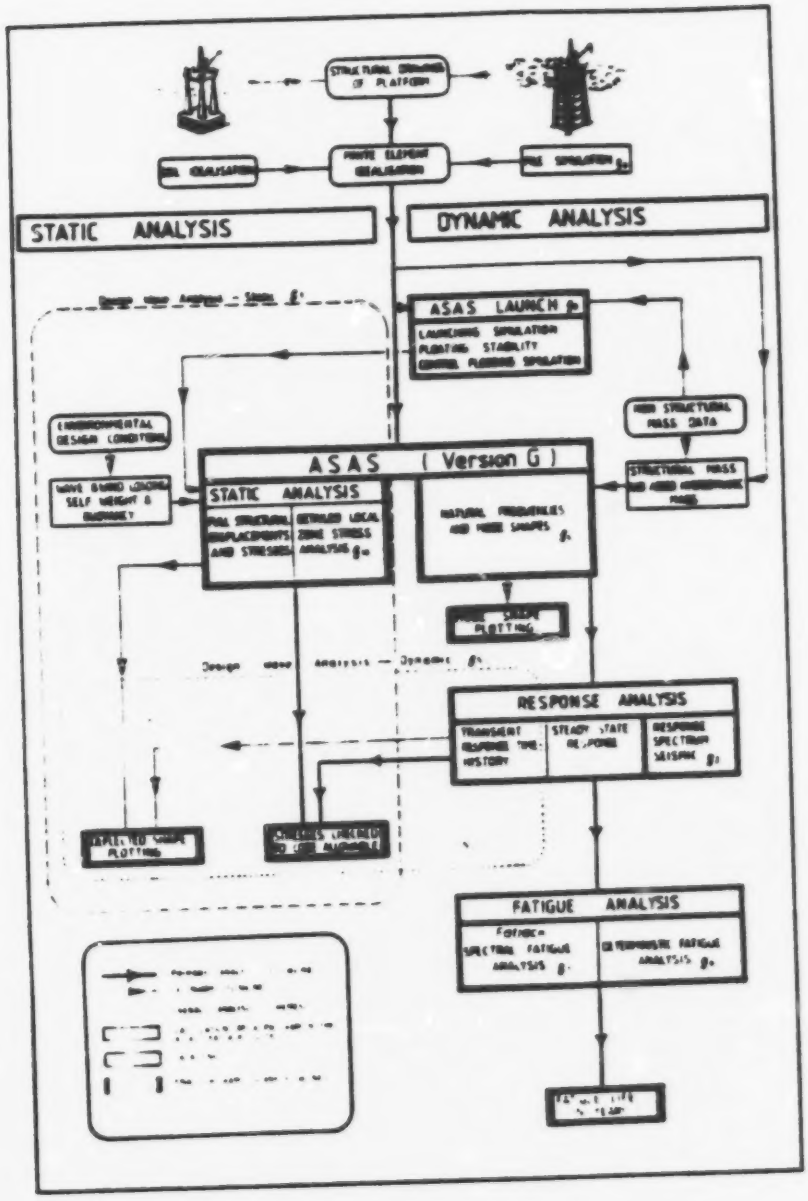


Fig. 3.6. ASAS: Offshore structural analysis system. (6703)

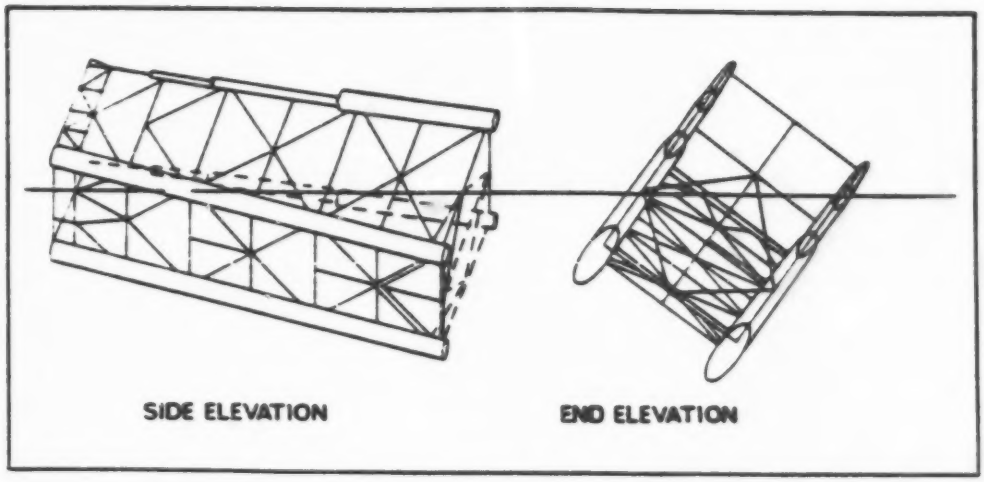


Fig. 3.7. Hydrostatic stability checks using ASASFLOAT. (neg 6698)

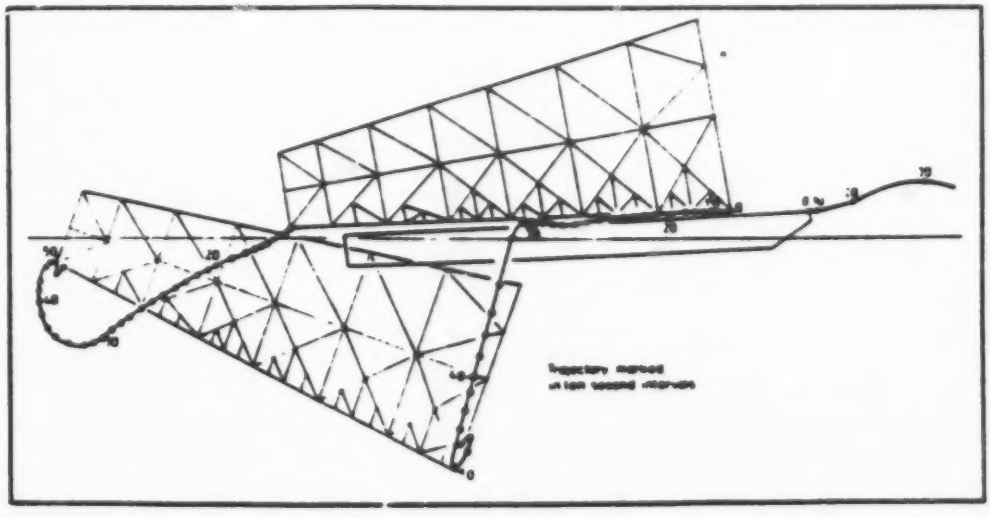


Fig. 3.8. ASASLAUNCH results for the launch trajectory of a steel jacket. (neg 6698).

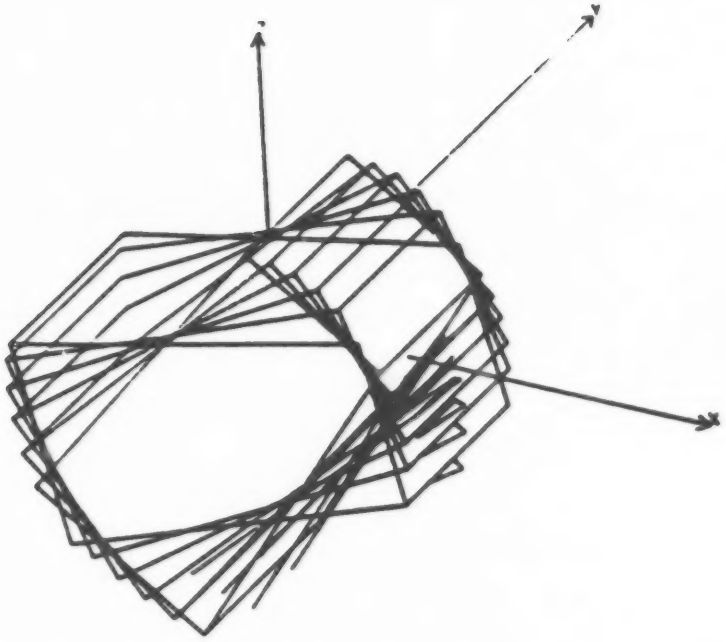


Fig. 3.9. The freely floating jacket oscillated until equilibrium is obtained. (neg 6704)

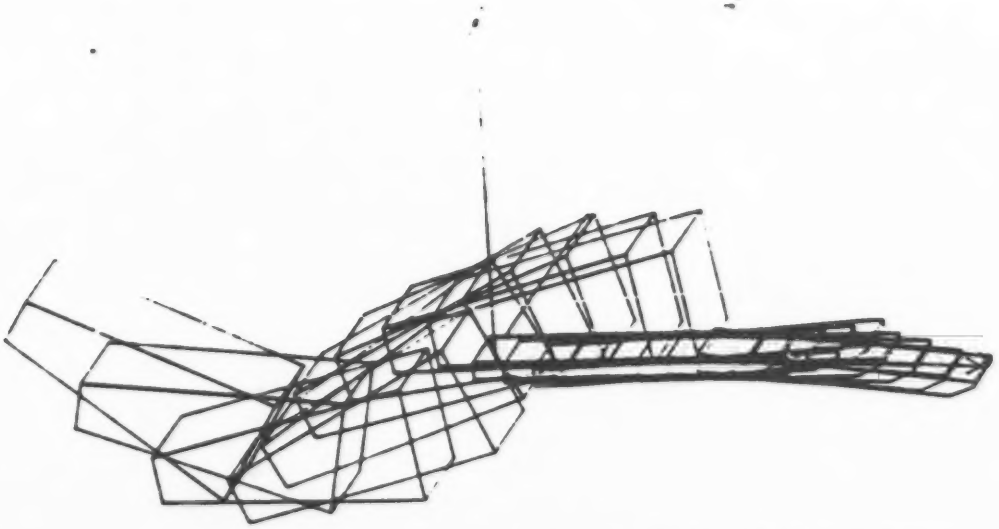


Fig. 3.10. Multiframe virtual image of jacket being launched from barge. (neg 6704)

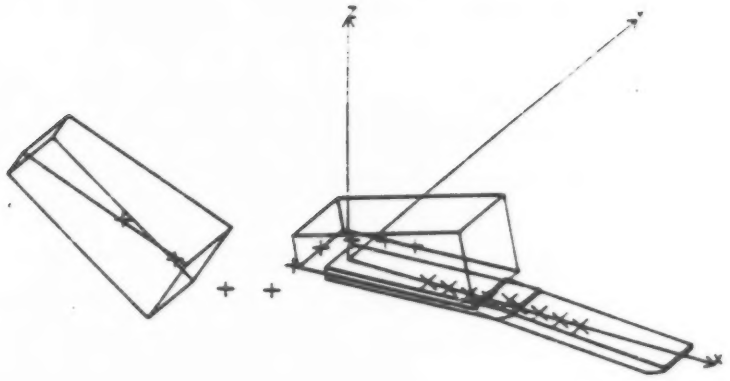


Fig. 3.11. The trajectories of the centres of gravity of barge and jacket. (neg 6714)

Fig. 3.12. Mesh generated using TUBEGEN. (neg 6700)

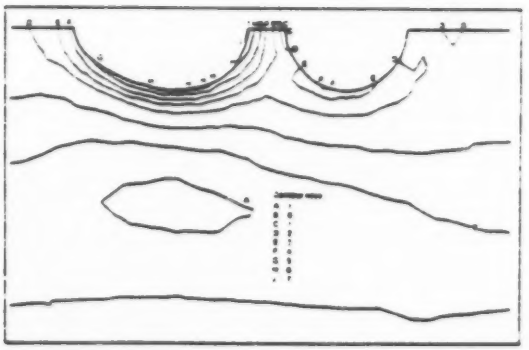


Fig. 3.13. Stress contour output: ASPECT. (neg 6700)

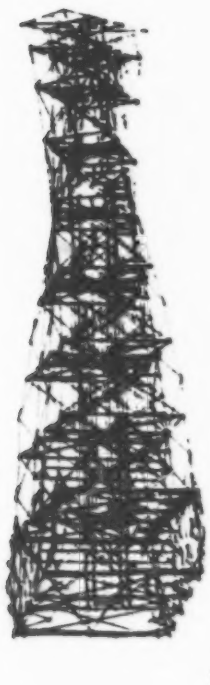


Fig. 3.14. Torsional mode shape as produced by STRAN. (neg 6707)

4. APPLICATION OF FE PROGRAMS TO THE ANALYSIS OF OFFSHORE STRUCTURES

4.1 INTRODUCTION

This chapter describes selected application of general purpose finite element programs for the solution of problems associated with offshore structures. This is not intended to be a comprehensive study of the literature, therefore, only selected publications will be described that demonstrate the use of the FE programs in the design of offshore structures.

Not every program described in Chapter 3 will be included because many of the analyses completed using the FE packages, e.g. ABS and SACS, are confidential to the clients concerned and are not published in the literature.

4.2 DNV/SESAM-69

4.2.1 Analysis of Joints

Gibstein (63) used the DNV/SESAM69 FE system to complete a parametric stress analysis study of tubular T joints. The S.C.F. were presented for joints loaded axially and for in-plane and out-of-plane bending. The influence of, the chord diameter to thickness ratio (γ), the thickness ratio (τ), the brace to chord diameter ratio (β), and the boundary conditions were investigated.

SCF formulae for T joints were developed from these numerical investigations, their accuracy was compared with experimental values, and found to be very good. These formulae are reported in section 2.4.5.

The calculations were performed using a doubly curved thin shell element contained in the program NV332. The tubular connections were modelled using the multilevel superelement technique in the program NV336. An example of the superelement representation is shown in Fig. 4.1.

The number of computer runs in such a parametric study, as described in (63), is of necessity very high and the elements in the area of the joint welds must be of sufficient refinement to give adequate numerical accuracy. Therefore, the use of superelement techniques reduces the data preparation and the computing costs by a substantial amount.

4.2.2 Analyses of CONDEEP Platform

Ramstad and Gudbrandsen (108) also reported the use of SESAM69 system to analyse the concrete gravity CONDEEP structure, Fig. 1.5. This problem was

too large for a straightforward analysis so again the superelement technique was used. The structure was modelled partly by 3-D solid elements and partly by shell elements. The structure was considered in two sections: the lower part consisted of the bottom slab, the cellar walls and lower domes; the upper part consisted of the upper domes and caisson region. Each of the two analyses considered one quarter of the structure and involved approximately 160,000 and 190,000 degrees of freedom respectively.

The lower structure analysis involved 10,000 elements and 19 loadcases and produced 85,000 pages of output. The upper section analysis consisted of approximately 12,500 elements and 37 loadcases and produced 215,000 pages of output. The output from the analyses was stored on magnetic tape for post-processing and the numerical results were presented in the form of 500 isocurve plots.

The numerical accuracy of the results for such a large computing problem was one of the main concerns of the investigators, and precautions were taken to ensure that rounding errors in the solution of the equations were kept within the limit of a few percent.

This series of analyses demonstrates the need for

(i) the capability, in FE programs, to divide a very large structure into portions of manageable size using substructures, and

(ii) the facilities for graphical presentation of numerical output; so that the analyses of large complex structures are economical in manpower and computing costs.

Other applications of the DNV/SESAM69 FE system are given in (109).

4.3 LLOYD'S/NASTRAN

4.3.1 Dynamic Analysis

Vugts et al (110) describe the dynamic analysis of a steel jacket platform using the Lloyd's NASTRAN package. The objective of this work was to perform both a technical and cost comparison of modal and direct dynamic analysis techniques. Three methods of analysis in the frequency domain were investigated using the NASTRAN dynamics package:

(i) a direct solution in physical coordinates

(ii) modal superposition using a varying number of modes

(iii) an alternative modal superposition technique supplemented by a static solution to include the response of higher modes.

The direct solution has not been exploited for the analysis of offshore platforms because, for large structures, this method requires powerful computing facilities capable of performing the required matrix operations efficiently. It is generally accepted that modal analysis is more economical and offers adequate accuracy particularly when supplemented by a static solution.

The model analysed was a realistic representation of a North Sea structure made up of 477 members having approximately 1500 degrees of freedom, and is shown in Fig. 4.2. The wave loading on the structure was a linear wave train of a 5s period represented by Lloyd's pre-processor for wave loads.

The Lloyd's NASTRAN dynamics package had the capability of performing the evaluation of the response to the prescribed wave excitation using modal superpositions of 10, 20, 30, 40 and 50 modes. Also these evaluations were repeated using the mode acceleration technique of NASTRAN, which includes higher modal components by the addition of a full static solution. This computer installation and package was also able to determine the dynamic response by a direct solution.

The authors concluded that:

(i) pure mode superposition was unacceptable if a realistic prediction of member forces and stresses is the primary objective

(ii) the enhanced modal acceleration technique offered a significant improvement in force and hence stress recovery

(iii) the direct solution offered an exact solution to the prescribed mathematical model, and

(iv) the relative cost comparisons showed that, contrary to accepted opinion, frequency domain direct solutions are cost effective compared with frequency domain modal solutions.

The authors therefore, recommended that whenever stress recovery is the primary objective and where direct solution facilities exist, a direct solution of the equations of motion should be performed.

4.3.2 Analysis of Stiffened Joints

Nicolussi et al (57) also used the NASTRAN analysis package for the static stress analysis of stiffened tubular joints. In this work the authors investigated Y joints with (i) no stiffeners, (ii) circumferential and longitudinal stiffeners inside the chord and (iii) the whole joint filled with concrete. The FE calculations were carried out using triangular and quadrilat-

eral bending elements to describe the joint and stiffeners and in the case of the filled joint solid tetrahedric elements were used.

From their work the authors concluded that

(i) the FE calculations based on automatic mesh generation and the analysis performed by NASTRAN was an easy, quick, cheap calculation method for tubular joints

(ii) the stiffened joints had stress levels three to five times lower than those for unstiffened joints

(iii) the concrete filled joints showed the lowest SCF of the joints analysed. Other applications of the NASTRAN program are described in (111) and (112).

4.4 ASAS

4.4.1 Analysis of Joints

One of the most important facilities on a FE analysis package is the capability of pre- and post-processing. Parkhouse (106) describes the mesh generation and results-presentation facilities of the ASAS package. The mesh generator described is the TUBEGEN program that generates input data for the analysis of tubular joints. This program is an interactive program which permits the analyst to generate the joint definition and a suitable FE mesh. The analyst can instruct the program to produce the node numbering and input data in a format acceptable to the analysis program when satisfied with the mesh produced. This procedure is a great time saver and relieves the operator of the tedium of producing large amounts of input data; also, the data produced will be error free. An example of the model and mesh produced for an overlapping K joint by TUBEGEN is shown in Fig. 3.12. The analysis was carried out in the normal way using quadrilateral elements.

The results produced by an FE analysis are voluminous and therefore a graphical presentation by the computer is essential. The author describes the ASAS post-processor ASPECT which produces contour plots of stress in the region of the joint as shown in Fig. 3.13. This method of presentation relieves the analyst of the wearisome study of large standard tabular printouts of deflections and stresses.

4.4.2 Dynamic and Fatigue Analysis

Hallam et al (113) describe the application of the ASAS FE system to the dynamic and fatigue analysis of the Murchinson Tower structure. This struc-

ture was installed in the North Sea in 156 m deep water in 1979. The FE model had 917 members and 376 nodes that involved 2235 degrees of freedom. The deck, tower and pile/soil foundation were modelled with tube, beam and axial spring elements. The added mass, flooded member mass and marine growth were handled automatically within the program ASASMASS. The structural steel mass was computed by ASAS DYNAMICS, and the wave and current loading was calculated for the model using ASASWAVE.

Two response analysis techniques were used,

(i) the transient solution for the design wave, where the time history response to a dynamic load was calculated

(ii) the steady state solution for the fatigue analysis.

The normal modes and natural frequencies were extracted from an ASAS DYNAMICS run and, in all, 25 modes were obtained. The storm wave response was determined using the transient response option of the ASAS-RESPONSE program. A storm wave from each of eight directions was considered and the loads were calculated for 10 different positions as the wave passed through the structure. The time history of the response on each of the normal modes was calculated.

The fatigue analysis was performed for 40 waves covering 8 directional sectors and the nodal forces were calculated for 10 different crest positions as each wave passed through the structure. The resulting time variation was characterised by a best-fit sine wave and a steady state response in each normal mode was calculated. The stress in terms of axial, bending and shear stress components was recovered from the displacement functions using the same techniques as in a static analysis. The total variation of stress was then calculated at each end of the tube member and combined with a SCF to determine the hot-spot stress range at eight points around the circumference of the tube. The SCF were calculated using the empirical formulae of Kuang et al (61). A Palmgren-Miner summation enabled the life to be predicted and compared with the target life of 30 years.

It was found that the dynamic storm wave analysis agreed well with the conventional static analysis which included dynamic amplification factors. The subsequent dynamic fatigue analysis showed several joints had inadequate life. Localised changes in the detail design of these joints were able to remedy this inadequacy.

Other applications of the ASAS package for the analysis of offshore structures are described in (114,115).

4.5 LAUNCH ANALYSIS COMPARISON

Few comparisons between the analysis capabilities for offshore structures of the FE packages described in this report were found in the literature. A report describing an investigation of the comparison between the launch analyses completed by ASASLAUNCH, MARCSLAUNCH and a scale model test will be examined (116). The jacket analysed had 1002 members and the model test had a scale factor of 1:82.67. The comparison was between the trajectories obtained as the jacket slipped off the barge, came to static equilibrium in the water and was finally up-ended.

The launch trajectory was considered to be two dimensional and was described by one rotation (θ) and two displacements (X,Z) of the jacket centre of gravity with respect to time, Fig. 4.3. The time dependence of these degrees of freedom is shown in Fig. 4.4 and the trajectory is punctuated by event markers:

0 - Winch force applied to stationary jacket.

1 - Rocker arm starts to rotate with respect to barge. Winch force released and jacket stops sliding relative to rocker arm tracks.

2 - Jacket overcomes static friction force and continues to slide relative to rocker arm tracks.

3 - Jacket separates from barge as trailing edge slides past rocker pin.

4 - Jacket displaces its weight of water and continues rotation to achieve a final position of static equilibrium.

The comparison of the trajectory as calculated by ASASLAUNCH, MARCSLAUNCH and the model test is given in Fig. 4.5. The main differences were found to be (i) in the time taken for the jacket to separate from the barge, (event 3) and (ii) in the angular velocity of the model during its final rotation to equilibrium which was significantly lower than for either computer analysis. The model test was initially in better agreement with ASASLAUNCH than with MARCSLAUNCH, but in the final rotation stage it did not agree with either computer analysis. The authors suggested several reasons for these differences namely: (i) in the computer analyses the hydrodynamic member loads were empirical and the boundary conditions could not be explicitly applied to predict relative movement between the jacket and barge and (ii) in the model test the hydrodynamic properties of the tubes were not modelled precisely for each tube, the static coefficient of friction was not modelled, and the scale of the model tubes created higher drag coefficients than were appropriate for the full scale structure.

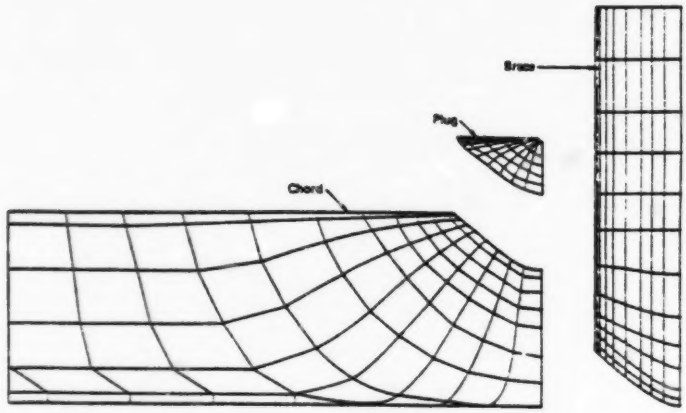


Fig. 4.1. FE mesh for T joint. (neg 6715)

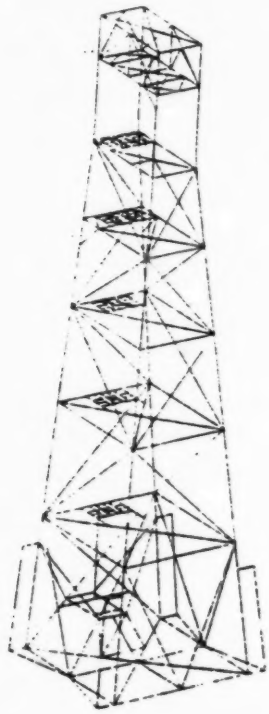


Fig. 4.2. Perspective line sketch of a jacket. (neg 6689)

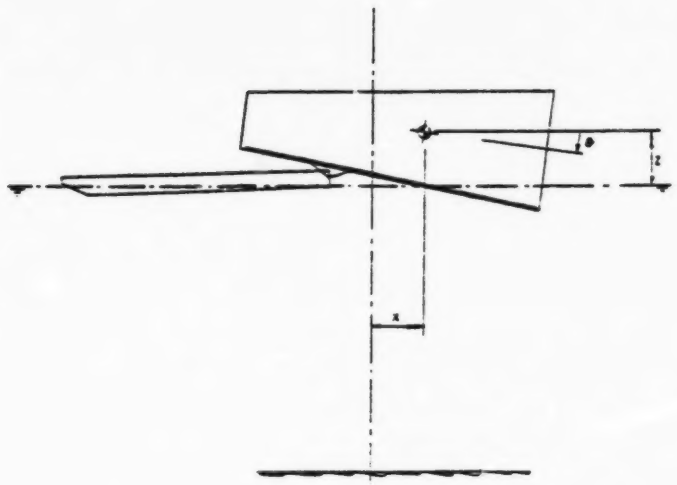
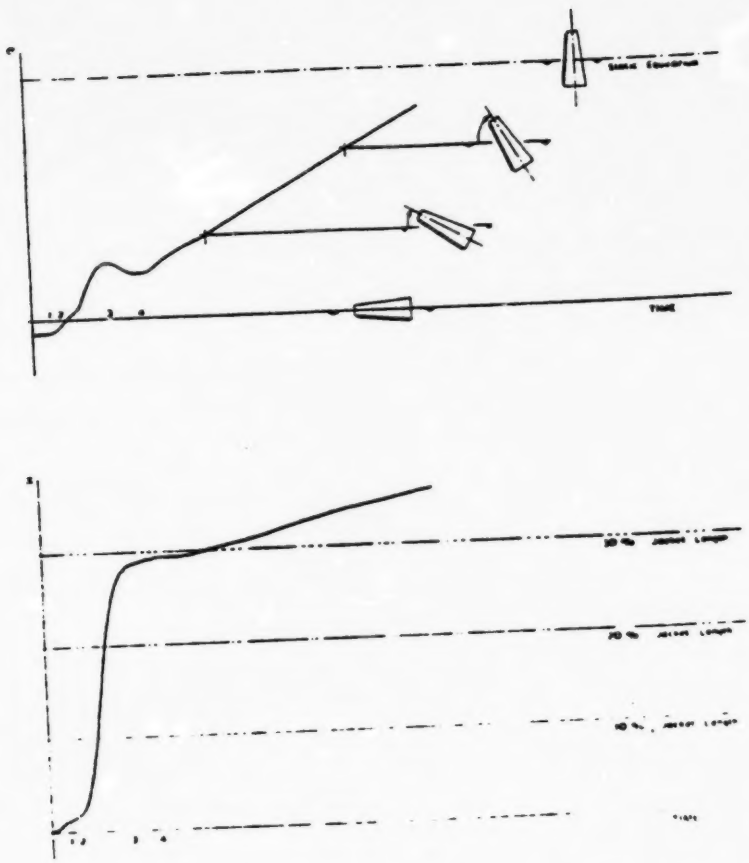


Fig. 4.3. Trajectory of jacket centre of gravity. (neg 6709)



(a) and (b). Guide to presentation of $\dot{\theta}$ and X trajectory. (neg 6705)

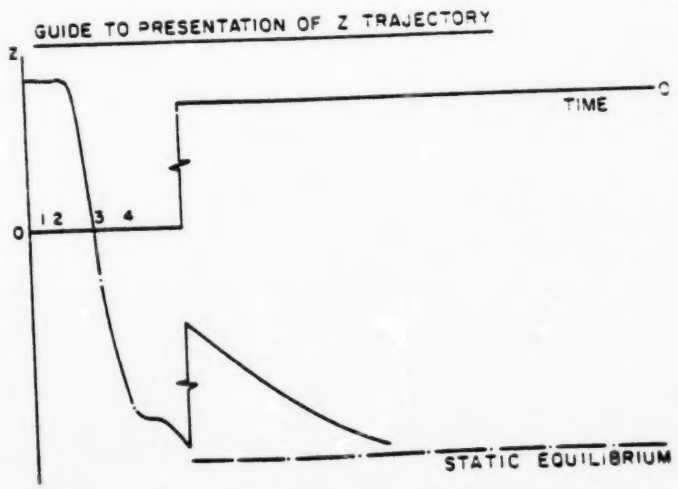
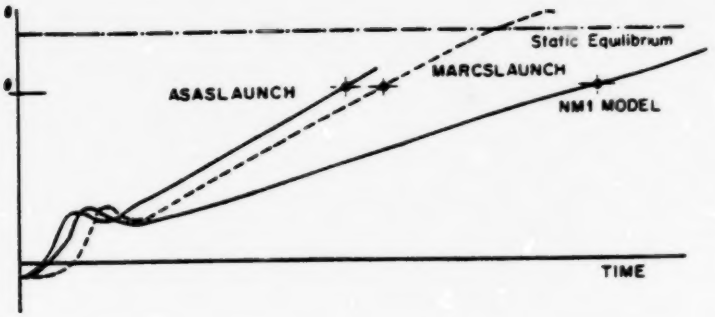
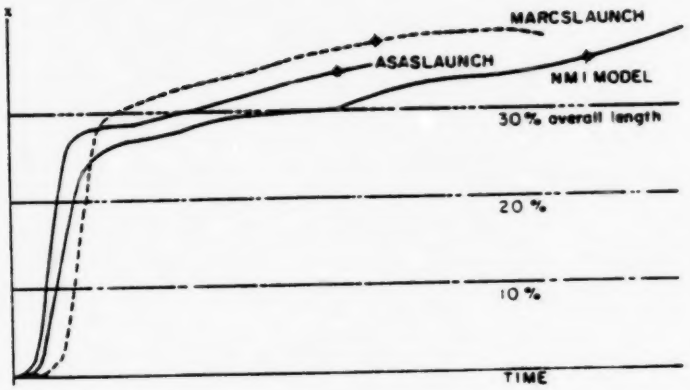


Fig. 4.4(c). Guide to presentation of Z trajectory. (neg 6729)





(a) and (b). θ and X trajectory computer analysis/model tests. (neg 6730)

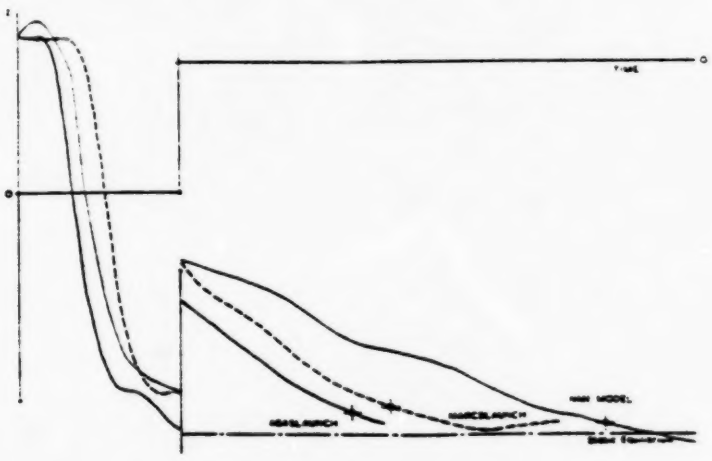


Fig. 4.5(c). Z trajectory computer analysis/model tests. (neg 6716)

Finally, the authors report a parametric study on the sensitivity of the trajectory obtained with variation of input parameters such as static and dynamic friction coefficients, hydrostatic characteristics and Reynold's number, and found that improvements in the trajectory calculated using ASASLAUNCH could be obtained.

5. CONCLUSIONS

1. The present designs and possible future developments in offshore exploration and production platforms have been described.
2. The design analyses required for modern offshore structures have been described.
3. The FE computer packages most widely used in the design of offshore structures have been identified. The capabilities of these programs have been described.
4. A list of references, a list of bureaux where the FE packages may be accessed, and a list of addresses of organizations where further information may be obtained, is included in the report.

6. REFERENCES

1. "The National Energy Program 1980"; (Report EP80-4E, Department of Energy Mines and Resources, Ottawa, Canada 1980).

2. "European Offshore Steels Research Seminar"; Cambridge 1978. (Published on behalf of the UK Dept. of Energy by the Welding Institute, Cambridge U.K. 1980).

3. "Steel in Marine Structures"; (Commission of the European Communities, Paris, France 1981).

4. Morrsion, A. "Cognac: World's Tallest Offshore Oil Platform"; (Civil Engineering, ASCE Vol. 50 N6 56-39; June 1980.

5. Harrison, J.D. "Material Selection Considerations for Offshore Steel Structures"; (Steel in Marine Structures, Paris. 148-193; 1981).

6. Marshall, P.W. "General Considerations of Tubular Joint Design"; (Paper 2, Welding in Offshore Structures, The Welding Institute Conference; February 1974).

7. Bell, A.O. and Walker, R.C. "Stresses Experienced by an Offshore Mobile Drilling Unit"; (Offshore Technology Conference, OTC 1440; 1971).

8. "The Alexander L. Kielland Accident" 1981; (Norwegian Petroleum Directorate; Oslo; 1981).

9. "One Piece Jacket Installed in Deep Gulf"; (Petroleum Engineer International; 12-15; Sept. 1981).

10. Ramstad, H. and Gudbrandsen, G. "The Analysis of the Condeep Stratfjord B Platform. Choice of Model with Regard to Wanted Accuracy, Time and Cost Involved"; (Computers and Structures, Vol. 12, 445-458; 1980).

11. Derrington, J.A. "Construction of McAlpine/Sea Tank gravity platforms at Ardyne Point, Argyll"; (Paper 13, Proc. of Conf. on Design and Construcion of Offshore Structures. Inst. of Civil Eng. 121-130; 1976).

12. Roren, E.M.Q. and Furnes, O. "State-of-the-Art: Behaviour of Structures and Structural Design"; (Marine Science Communications 4(3), 179-220; 1978).

13. Lalli, D. "Design, construction and installation of the Loango steel gravity platforms"; (Design and Construction of Offshore Structures, Proc. of a Conf. Held 1976 Institution of Civil Engineers, London; 1976).

14. Strickland, G.E. and Mason, A.B. "Parametric Response of TLP Tendons - Theoretical and Numerical Analyses"; (Offshore Technology Conference, OTC 4071, 45-54; Houston; 1981).

15. Jefferys, R.E. and Patel, M.H. "Dynamic Analysis Models of the Tension Leg Platform"; (Offshore Technology Conf. OTC 4075, 99-108; Houston; 1981).

16. Perrett, G.R. and Webb, R.M. "Tethered Buoyant Platform Production System"; (Offshore Technology Conf. OTC 3881, 261-273; Houston; 1980).

17. Mercier, J.A., Goldsmith, R.G. and Curtis, L.B. "The Hutton TLP; A Preliminary Design"; (European Offshore Petroleum Conference and Exhibition, EUR 264, 525-533; London; 1980).

18. Mangiavacchi, A., Abbott, P.A., Hanna, S.Y and Suhendra, R. "Design Criteria of a Piled Founded Guyed Tower"; (Offshore Technology Conf. OTC 3882, 275-286; Houston; 1980).

19. Mes, M.J. "Guyed Tower Design and Analysis"; (Petroleum Engineer International; Part I - The Concept; Nov. 1981; Part II - Static Analysis of Preliminary Guy Design, Nov. 15, 1981; Part III - Preliminary Calculation Procedures, Dec. 1981; Part IV - Designing for Large Deflection and Other Non-linear Effects, Jan. 1982; Part V - Final Dynamic and Fatigue Analysis, Feb. 1982).

20. Leidersdorf, C.B. and Potter, R.E. "Slope Protection for Artifical Exploration Islands off Purdhoe Bay"; (Offshore Technology Conf. OTC 4112, 437-447; Houston; 1981).

21. Chari, T.R. and Peters, G.R. "Engineering Challenges in the Cold Oceans"; (J of the Soc. for Underwater Technology; 8-13; March 1981).

22. Wetzel, V.F., Berenger, D.M. and Jozan, M.M. "Potential Schemes for Offshore Labrador Year Round Production"; (Offshore Technology Conf. OTC 3711, 293-305; Houston; 1980).

23. Boaz, I.B. and Bhula, D.N. "A Steel Production Structure for the Alaskan Beaufort Sea"; (Offshore Technology Conf. OTC 4113, 449-459; Houston; 1981).

24. Chapman, J.C. "Design and Construction of Offshore Structures"; (Proc. of the 2nd International Conference on the Behaviour of Offshore Structures, BOSS'79, 59-72; London; 1979).

25. Cowan, J.B. "Certification - The First Five Years"; (European Offshore Petroleum Conference and Exhibition, EUR 240, 273-277; London; 1980).

26. American Bureau of Shipping. "Guide for Building and Classing Fixed Offshore Structures"; 1978.

27. Department of Energy U.K. "Offshore Installations: Guidance on Design and Construction"; (HMSO; London; 1977).

28. Lloyd's Register of Shipping. "Rules for the construction and classification of mobile offshore units"; 1972.

29. Det Norske Veritas. "Rules for the Construction and Classification of Mobile Offshore Units"; 1975.

30. American Petroleum Institute. "Recommended Practice for Planning, Designing and Constructing Fixed Offshore Platforms"; (API RP2A, Twelfth Edition; 1981).

31. Roren, E.M.Q. and Furness, O. "State-of-the-Art: Behaviour of Structures and Structural Design"; (Marine Science Communications, 4(3), 179-220; 1978).

32. Brebbia, C.A. "Analysis of Offshore Structures"; (Offshore Structures Engineering, Conference at Rio de Janeiro; 1977).

33. Brebbia, C.A. and Walker, R.C. "Dynamic Analysis of Offshore Structures"; (Newnes-Butterworths, London; 1979).

34. Darbyshire, M. and Draper, L. "Forecasting Wind Generated Sea Waves"; (Engineering 5, 195, 482-484; April 1963).

35. Hogben, N. "Fluid Loading on Offshore Structures, a State-of-the-Art Appraisal: Wave Loads"; (Rev. Inst. of Naval Arch.; 1974).

36. Morison, J.R. et al. "The Force Exerted by Surface Waves on Piles"; (Petrol. Trans., Am. Inst. Mining Metal Engineering; 1950).

37. Dean, R.G. "Relative Validities of Water Wave Theories"; (Proc. ASCE, Waterways and Harbours Div. 96 WW1; 1970).

38. Berkhoff, J.C.W. "Computation of Combined Refraction - Diffraction"; (Conf. on Coastal Engineering. Inst. of Civil Engineers; 1973).

39. Hogben, N. and Standing, R.G. "Wave Loads on Large Bodies"; (Proc. Int. Symposium on Dynamics of Marine Vehicles and Structures in Waves. Inst. of Mech. Engrs.; London; 1974).

40. "Wave Loading on Offshore Structures - Theory and Experiment"; (Proc. Symposium on Ocean Engineering. Royal Inst. Naval Architects; 1974).

41. "A Critical Evaluation of Data on Wave Force Coefficients"; (British Ship Research Association Report).

42. Hogben, N., Miller, B.L., Searle, J.W. and Ward, G. "Estimation of Fluid Loading on Offshore Structures"; (National Maritime Institute Report No. R11).

43. "Boss'76. Proceedings of the International Conference on the Behaviour of Offshore Structures"; (Trondheim; 1976).

44. "Boss'79. Proceeding of the Second International Conf. on the Behaviour of Offshore Structures"; (London; 1979).

45. Schum, W. "The Structural Design Appraisal of Deepwater Steel Platforms"; (Offshore Structures Engineering Ed. Canecro, F.L. et al., Vol. 1; Gulf Pub.; 1979).

46. Hasselmann, K. et al. "Measurements of wind-wave growth and swell decay during the Joint North Sea Wave Project (JONSWAP)".

47. Pierson, W.J. and Moskowitz, L.A. "A proposed spectral form for fully developed wind seas based on the similarity theory of S.A. Kilaigorodskii"; (J Geophys. Res., 69, 5181-5190; 1964).

48. Nair, D., Valivieso, J.B. and Johnson, C.M. "Comparison of Spectrum and Time History Techniques in Seismic Design of Platforms"; (Offshore Technology Conference, OTC 3823; 1980).

49. "European Offshore Steels Research Project"; (UK Department of Energy, Cambridge, England; 1978).

50. "Steel in Marine Structures"; (Commission of the European Economic Communities, Paris; 1981).

51. Marshall, P.W. "General Considerations for Tubular Joint Design"; (Proc. of Intl. Conf. on Welding in Offshore Constructions, Newcastle-upon-Tyne, England; Paper 2; 1974).

52. American Welding Society - structural welding code; (AWS DI 1-72).

53. Wordsworth, A.C. and Smedley, G.P. "Stress Concentrations at Unstiffened Tubular Joints"; (European Offshore Steels Research Seminar; Paper 31; Cambridge, U.K.; 1978).

54. Clayton, A.M. and Irvine, N.M. "Stress Analysis Methods for Tubular Connections"; (European Offshore Steels Research Seminar; Paper 30; Cambridge, U.K.; 1978).

55. Fessler, H., Little, W.J. and Shelland, I.J. "Stresses in Tubular K Joint Subjected to Out-of-Plane Bending"; (European Offshore Steels Research Seminar; Paper 29; Cambridge, U.K.; 1978).

56. UKOSRP - Final report to ECSC (F7); 1981.

57. Nicolussi, F., Gnone, E. and Sebastiani, "Stress Analysis of Stiffened Tubular Joints; (European Offshore Steels Research Seminar; Paper 27; Cambridge, U.K.; 1978).

58. Pozzolini, P.F. "Tests on Tubular Joints"; (International Conf. on Steel in Marine Structures - Special and Plenary Sessions; 27; Paris; 1981).

59. De Back, J. "Strength of Tubular Joints"; (Intl. Conf. on Steel in Marine Structures - Special and Plenary Sessions; 365; Paris; 1981).

60. Radenkovic, D. "Stress Analysis in Tubular Joints"; (Intl. Conf. on Steel in Marine Structures - Special and Plenary Sessions; 53; Paris; 1981).

61. Kuang, J.G., Potvin, A.B., Leick, R.D. and Kahlich, J.L. "Stress Concentration in Tubular Joints"; (SPE Journal; 287-299; 1977).

62. Kuang, J.G., Potvin, A.B. and Leick, R.D. "Stress Concentration in Tubular Joints"; (Offshore Technology Conf.; Paper OTC 2205; 1975).

63. Gibstein, M.B. "Parametric Stress Analysis of T Joints"; (European Offshore Steels Research Seminar; ;Paper 26; Cambridge; 1978).

64. Williams, A.K. and Rinne, J.E. "Fatigue Analysis of Steel Offshore Structures"; (Proc. Inst. Civil Engrs.; Part 1; 60; 635-654; 1976).

65. Maddox, N.R. "Fatigue Analysis for Deepwater Fixed Bottom Platforms"; (Offshore Technology Conf.; Paper OTC 2051; 1974).

66. Maddox, N.R. and Wildenstein, A.W. "A Spectral Fatigue Analysis for Offshore Structures"; (Offshore Technology Conf.; Paper OTC 2261; 1975).

67. Kallaby, J. and Price, J.B. "Evaluation of Fatigue Considerations in the Design of Framed Offshore Structures"; (Offshore Technology Conf.; Paper OTC 2609; 1976).

68. Smedley, G.P. "The Avoidance of Fatigue Failures in Offshore Steel Structures"; ((Int. Inst. of Welding) Public Sess. and Met. Technol. Conf.; Sydney, Aust.; VB; Sess 17; Paper 3; 37; 1976).

69. "Report of the Inquiry into the Causes of the Accident to the Drilling Rig Sea Gem"; (HMSO; London; 1967).

70. Bell, A.O. and Walker, R.C. "Stresses Experienced by an Offshore Mobile Drilling Unit"; (Offshore Technology Conf.; Paper OTC 1440; 1971).

71. Palmgren, A. "Die Lebensdouen von Kugelhagern"; (VDIZ; 68; 14; 339; 1924).

72. Miner, M.A. "Cumulative damage in fatigue"; (J. Appl. Mech.; 12; A-159; 1945).

73. Cronin, D.J., Godfrey, P.S., Hook, P.M. and Wyatt, T.A. "Spectral Fatigue Analysis for Offshore Structures"; (Numerical Methods in Offshore Engineering; Ed Zienkiewicz and O.C. Wiley; 1978).

74. Gurney, T.R. and Maddox, S.J. "A Reanalysis of Fatigue Data for Welded Joints"; (Welding Institute Report E/44/72; 1972).

75. Snedden, N.W. "Offshore Installations: Guidance on Design and Construction. Proposed New Fatigue Design Rules for Steel Welded Joints in Offshore Structures"; (U.K. Department of Energy; June 1981).

76. Bristoll, P. "A Review of the Fracture Mechanics Approach to the Problems of Design, Quality Assurance, Maintenance and Repair of Offshore Structures"; (European Offshore Steels Research Seminar; Paper 24; Cambridge, U.K.; 1978).

77. Schutz, W. "Procedures for Prediction of Fatigue Life of Tubular Joints"; (Intl. Conf. on Steel in Marine Structures - Special and Plenary Sessions; 254; Paris; 1981).

78. Dover, W.D., Hibberd, R.D. and Holdbrook, S.J. "A Fracture Mechanics Analysis of Fatigue Failure of T-Joints Subject to Random Loading"; (Intl. Symp. - "Integrity of Offshore Structures"; Glasgow; 1978).

79. Tomkins, B. and Scott, P. "An Analysis of the Fatigue Endurance of Tubular T-Joints by Linear Elastic Fracture Mechanics"; (European Offshore Steels Research Seminar; Paper 20; Cambridge, U.K.; 1978).

80. Paris, P.C. "The Growth of Cracks Due to Variations in Load"; (PhD Thesis; Lehigh University; 1960).

81. Foreman, F.G., Kearney, V.E. and Engle, R.M. "Numerical Analysis of Crack Propagation in Cyclic Loaded Structures"; (J of Basic Engineering; 459-464; 1967).

82. Klesnil, M. and Lukas, P. "Influence of Strength and Stress History on Growth and Stabilisation of Fatigue Cracks"; (Engineering Fracture Mechanics; Vol. 4; 77-92; 1972).

83. Zienkiewicz, O.C. "The Finite Element Method in Engineering and Science"; (McGraw-Hill; London; 1971).

84. Gallagher, R.H. "Finite Element Analysis Fundamentals"; (Prentice-Hall; 1975).

85. Clough, R.W. and Penzien, J. "Dynamics of Structures"; (McGraw-Hill; 1975).

86. Fredriksson, B. and Mackerle, J. "Structural Mechanics Finite Element Computer Programs, Surveys and Availability"; (Linkoping Institute of Technology, Div. of Solid Mechanics; Linkoping, Sweden; 1977).

87. Noor, A.K. "Survey of Computer Programs for Solution of Nonlinear Structural and Solid Mechanics Problems"; (Computers and Structures; Vol. 13; 425-465; 1981).

88. Robinson, J. "Integrated Theory of Finite Element Methods"; (John Wiley and Sons; 1973).

89. Egeland, O. and Araldsen, P.O. "SESAM-69 - A General Purpose Finite Element Method Program"; (Computers and Structures; Vol. 4; 41-68; 1974).

90. "SESAM-69 General Description"; (D.N.V. Computas Technical Report 78-921; 1979).

91. "Engineering Programs. A Survey of Engineering Programs of Det Norske Veritas and AS Computas"; (AS Computas, Høvik, Norway; 1977).

92. "MSC/NASTRAN Handbook for Linear Static Analysis"; (MSC/NASTRAN Version 60; The MacNeal-Schwendler Corp.).

93. "MSC/NASTRAN Dynamic Analysis"; (The MacNeal Schwendler Corp.)

94. "Offshore Services Brochure"; (Lloyd's Register of Shipping; 1979).

95. "Computer Services of Lloyd's Register"; (Lloyd's Register of Shipping; 1979).

96. Short, J.D. and Cobern, P.J.W. "LR LOPS: A Computer System for the Analysis of Fixed Steel Offshore Structures"; (CAD 80: Int. Conf. and Exhib. on Comput in Des. Eng. 4th; Brighton Sussex, England; 446-464; 1980).

97. Nataraja, R. "Finite Element Methods in the Design Appraisal of Offshore Platforms"; (Finite Element News; Robinson and Associates; England; Sept. 1979).

98. "ABS Engineering Computer Programs"; (American Bureau of Shipping, Ocean Engineering Division; Computer Program Document OE-5084; 1981).

99. Perrone, N. and Pilkey, W. (Editors) "Structural Mechanics Software Series"; Vo. II; (University Press of Virginia; 1978).

100. Pilkey, W.D., Saczalski, K.J. and Schaeffer, H.G. "Structural Analysis Computer Programs"; (University Press of Virginia; 1974).

101. "Structural Analysis Computer System"; (Engineering Dynamics Inc.; New Orleans).

102. ASAS User Manual - Version G; (Atkins Research and Development; Epsom, England).

103. ASAS Ancillary Programs; (Atkins Research and Development; Epsom, England; 1981).

104. "Analysis of Offshore Structures with ASAS/FATJACK Related Programs"; (Atkins Research and Development; Epsom, England; 1979).

105. ASASLAUNCH, ASASFLOAT, C-gul; (Atkins Research and Development; Epsom, England).

106. Parkhouse, J.G. "Computer-Aided Analysis of Tubular Joints"; (European Offshore Steels Research Seminar; Paper 28; Cambridge; 1978).

107. Kacprzynski, J.J. "Finite Element Programs for Aerospace Structural Analysis"; (Technical Report LTR-ST-932; National Research Council, Ottawa, Canada; 1977).

108. Ramstad, H. and Gudbrandsen, G. "The Analysis of the CONDEEP Stratfjord B Platform. Choice of Model with Regard to Wanted Accuracy, Time and Cost Involved"; (Computers and Structures; Vol. 12; 445-458; 1980).

109. Gibstein, M.B. and Moe, E.T. "Numerical and Experimental Stress Analysis of Tubular Joints with Inclined Braces"; (Steel in Marine Structures; Paper 6.3; Conf. Proc.; Paris; 1981).

110. Vugts, J.H. et al "Modal Superposition vs Direct Solution Techniques in Dynamic Analysis of Offshore Structures"; (Boss'79; 2nd Int. Conf. Behaviour of Offshore Structures; Paper 49; London, England; 1979).

111. Gantke, F. "Design Concept of Cast Joints"; (Steel in Marine Structures; Paper 4.5; Conf. Proc.; Paris; 1981).

112. Perrett, G.R. and Webb, R.M. "Tethered Buoyant Platform Production System"; (Offshore Technology Conf. OTC 3881; 1980).

113. Hallam, M.G. et al "Dynamic and Fatigue Analysis of Murchinson Tower Structure"; (Offshore Technology Conference, OTC 3163; Houston; 1978).

114. Cannell, C. and Starsmore, N. "A Retrospective Assessment of Fatigue Damage Suffered by West Sole Platform W.E."; Houston; (Offshore Technology Conference, OTC 3904; 1980).

115. Spooner, J.B. and Wooton, L.R. "Structural Response of Offshore Structures"; (Meerestech. Mar. Technol. Vol. 8; n3; 93-98; 1977).

116. Lyons, R.A. and Smith, S.L. "Launch Analysis Comparison"; (Atkins Research and Development; Epsom, England; 1978).

APPENDIX I

ADDRESSES

1. Det Norske Veritas
   Suite 1001, 800 Place Victoria
   Montreal, H4Z 1A2
   Telephone: (514) 866-9336
   Contact: Hans Haakonsen

2. Det Norske Veritas
   AS Computas
   P.O. Box 310
   1322 Høvik
   Norway

3. Lloyd's Register of Shipping
   400 St. Antoine W
   Montreal
   Telephone: (514) 849-4291

4. Lloyd's Register of Shipping
   Offshore Services Group
   71 Fenchurch Street
   London, EC3M 4BS

5. American Bureau of Shipping
   360 St. James
   Montreal
   Telephone; (514) 844-3029

6. American Bureau of Shipping
   65 Broadway
   New York, NY 10006
   Telephone: (212) 440-0300
   Contact: Faith K. Lee (440-0470)

7. SACS:
   Engineering Dynamics Inc.
   1001 Howard Street Suite 2400
   New Orleans, Louisiana, 70113
   Telephone: (504) 522-3506
   Contact: Jim Angehr

8. ASAS:
   Atkins Research and Development
   Woodcote Grove, Ashley Road
   Epson, Surrey, KT18 5BW
   England

9. ASAS (Canada)
   H.G. Engineering
   260 Lesmills Road
   Don Mills, Ontario
   Telephone: (416) 447-5535
   Contact: Dr. Tony Firmin

10. MARCS:
    Synercom Technology Inc.
    P.O. Box 27
    500 Corporate Drive
    Sugar Land, Texas 77478
    Telephone: (713) 491-5000

11. NASTRAN:
    The MacNeal-Schwendler Corp.
    Midwest Regional Office
    5745 Oxford Drive
    New Berlin, Winconsin
    53151
    Telephone: (414) 542-5747
    Contact: Michael J. Krauski

12. American Petroleum Institute (API)
    156 William
    New York, NY
    Telephone: (212) 587-9660

13. Multiple Access
    785 Carling Ave. Suite 704
    Carling Square
    Ottawa, Ontario K1S 5H4
    Telephone: (613) 238-8583
    Contact: Bruce Mason

14. Control Data Canada Ltd.
    130 Albert Street, Suite 1105
    Ottawa, Ontario K1P 5G4
    Telephone: (613) 238-2325
    Contact: Malcolm Harrison

POOR

QUALITY

ORIGINAL